Perceptive DataTransfer

Administrator Guide

Version: 7.0.x

Written by: Documentation Team, R&D
Date: September 2020

Hyland

Copyright

Hyland

Table of Contents

Introduction

Perceptive DataTransfer is a data matching, review and upload solution that allows you to automatically verify, clean up, and enter data extracted from hand-printed paper forms, downloadable web forms, and almost any other electronic source.

You can quickly and accurately upload information into your database, such as online and paper applications, recruit cards, SAT scores, ACT scores, GRE scores, and AP scores without the risk of creating duplicate student records.

With Perceptive DataTransfer, you can:

- Import a nearly unlimited range of data files.
- Match input data to records in your database.
- Review potentially matching records side-by-side.
- Upload new or modified information in real-time into your database.

You can customize Perceptive DataTransfer to upload any type of data from any information source in the Perceptive DataTransfer source code.

Refer to the following topics for more information.

Starting and Logging in to Perceptive DataTransfer

To start and log in to Perceptive DataTransfer, complete the following steps.

1. Start the application by doing one of the following:
    - Open a web browser and type the Perceptive DataTransfer location into the browser address bar. For example, type an address such as http://*servername*:8080/DataTransfer.
    - On desktop computers, open the Perceptive DataTransfer location by selecting File > Open from the browser menu and provide the location.

- On desktop computers, double-click the Perceptive DataTransfer shortcut icon, or (in Microsoft Windows) right-click it and click **Open**, or (in Apple Mac and Linux) control-click it and click **Open**.

  By default, there is no shortcut icon. On desktop computers, you can create a desktop shortcut, or, for tablets, phones, and desktop computers, you can bookmark the startup page to quickly access it when you subsequently log in to the application.

  The log in screen appears.

2. When you log into Perceptive DataTransfer for the first time, you are prompted to accept the End User License Agreement (EULA). Click **Accept** to accept the agreement. When you accept the agreement for your username, it is also accepted for all other roles to which you belong; if you switch to another Role, you do not have to accept the EULA again. If you click **Decline**, you are redirected to the log in screen.
3. Type your username and password in the **Username** and **Password** fields. Your username and password are usually the same as those you use to log in to your administrative system.

   Note If there are issues starting Perceptive DataTransfer or entering data in the login screen, contact your Perceptive DataTransfer system administrator for assistance in establishing the necessary connection rights and privileges.
4. Click **Log in**.

   If you are logging into a workspace version in **Edit** mode and another user is logged into that version of the workspace, you receive a prompt that the **Sportin**version is locked by another user.

   You are logged in to Perceptive DataTransfer, and the application starts.

   You are logged in to Perceptive DataTransfer with your default user role and workspace version in which you are working. Specific product functions and features are available depending on the role you use and on the workspace version in which you are working.

   Note: After 25 minutes of inactivity, you are prompted that you will be logged out within five minutes of further inactivity.

Understanding the Desktop and Tablet Interfaces

Perceptive DataTransfer displays the desktop view when you use the application on desktop computers; it displays the tablet view when you use it on tablets and on phones with larger resolutions. The layout differs slightly between the interfaces, and not all features are available for the tablet view.

The Job Dashboard displays all jobs that are configured for Perceptive DataTransfer and allows users to run jobs in batch mode.

The Perceptive DataTransfer toolbar provides access to Perceptive DataTransfer features. Refer to the following table for a description of the toolbar menu options.

Element	Description
Perceptive button	Contains the following options. • Open Admin Guide: Provides access to the administrator guide. • Open User's Guide: Provides access to the user guide. • About DataTransfer: Provides information about Perceptive DataTransfer. • Licensing: Provides licensing information.

Element	Description
Jobs button	Contains the following options. • Add Job: Allows you to add jobs. • Recurring Jobs: Allows you to add recurring jobs. Refer to Configuring Jobs and Processing Records/
Tools	Contains the following options. • Lookup: Allows you to configure the lookup tool. • Check Names: Allows you to configure the check names tool. • Data Exports: Allows you to configure data exports. • Logs: Allows you to view log messages about jobs. • Import History: Allows you to view details about workspace imports. Refer to Using Perceptive DataTransfer Tools.
Configuration	Contains the following options: • Connections: Allows you to configure database connections. Refer to Configuring connection groups and database connectors. • File sources: Allows you to configure servers or SFTP sites that contain the input file used for recurring jobs. Refer to Configuring File Sources. • Procedures: Allows you to configure procedures. Refer to Configuring Procedures. • Headers: Allows you to configure headers. Refer to Configuring headers. • Forms: Allows you to configure forms. Refer to Configuring Forms. • Notifications: Allows you to configure notifications to be sent when various activities occur. For more information, refer to Configuring Notifications. • ImageNow: Allows you to configure ImageNow index maps. For more information, refer to Configuring ImageNow Index Maps. • Data Providers: Allows you to configure data providers. Refer to Configuring data providers. • Document Templates: Allows you to create document templates to be used in NQL code. For more information, refer to Configuring Document Templates. • Variable Maps: Allows you to create variable mappings to map variables to labels and default values. For more information, refer to Configuring Variable Maps. • EDI formats: Allows you to create EDI formats. Refer to Configuring EDI formats.

Element	Description
Reports	Contains the following options. • Configure: Allows you to configure reports. • Schedule: Allows you to configure scheduled reports. • View: Allows you to view reports. For more information, refer to Configuring Reports.
Preferences	Allows you to configure Perceptive DataTransfer preferences; it contains the following options. • Set Default Role: Sets the current user's role as the default role. • Set Default Version: Sets the workspace version of the current user as the default version. • Record Grid Dimensions: Specifies the number of rows and columns per page to display for records returned by a job. • Filename in Job Name: Appends the filename of the input file to the name of a job when you add jobs. • Verify before running job: Confirms that you want to run the job. • Display All Match Results: When jobs are run in interactive mode, all possible record matches are compiled before the application pauses to allow you to review the possible matches. • Skip Previous Results: Prevents possible match records from being displayed twice. Refer to Specifying Preferences.
Administration	Contains the following options. • Workspace: Allows you to configure workspaces. Refer to Changing Workspace Versions. • User: Allows you to configure user and administrator privileges. Refer to Configuring Users and Privileges. • Share Groups: Allows you to configure share groups, which you use to share workspaces and workspace objects. Refer to Changing Workspace Versions. • Server Logs: Allows you to view server logs. Refer to Viewing Server Logs. • Archive: Allows you to configure rules for archiving jobs. Refer to Configuring Jobs and Processing Records. • System Properties: Contains the following options. ◦ System Email: Allows you to configure the SMTP server from which email notifications are sent. For more information, refer to Configuring Notifications. ◦ PDF to TIFF: Allows you to configure properties for PDF to TIFF file conversion. For more information, refer to Configuring PDF to TIFF Properties.

Element	Description
Role drop-down box	Allows you to change the role into which you are logged into Perceptive DataTransfer. Refer to Changing Roles.
Workspace drop-down box	Allows you to change the workspace version that you are using. Refer to Changing Workspace Versions.
Logout button	Logs you out of the application.

If you are logging into Perceptive DataTransfer using phones with smaller screens and resolutions, the mobile Job Dashboard is displayed; the exact design varies among devices, but the layout and functionality remain the same. For more information about working with jobs, refer to Configuring Jobs and Processing Records.

Sorting Columns

For Perceptive DataTransfer dialog boxes or windows that contain tables of information, you can choose the columns of information you want to view or hide, sort information alphabetically, and resize the width of columns.

To select the columns that display and to sort columns, complete the following steps.

1. In the dialog box, point your mouse to the edge of a column and click .
2. Click **Sort Ascending** to sort information alphabetically in a high to low sequence (A to Z).
3. Click **Sort Descending** to sort information alphabetically in a low to high sequence (Z to A).
4. Click **Columns**.
5. Check the check box for the column you want to display.

Resizing Columns

To resize the width of a column, point your mouse over the edge of a column until the mouse pointer changes to ; then, drag and drop your mouse to the desired location to resize the column.

Expanding Panes

You can collapse and expand some panes. For panes on the right side of the application window, click to collapse the pane click to expand a collapsed pane; for panes on the left side of the application window, click to collapse the pane and to expand the pane.

Refreshing Information

To refresh information in dialog boxes and application windows, click .

Changing Roles

Perform one of the following actions.

- If you are using the desktop version of DataTransfer, from the Role drop-down box, select the Perceptive DataTransfer role you want to use.
- If you are using the mobile version of DataTransfer, select **Settings > Role**, and then select the role you want to use.

Changing Workspace Versions

Perform one of the following actions.

- If you are using the desktop version of DataTransfer, from the Workspace drop-down box, select the version you want to use.

  When you hover your mouse over a version in the drop-down box, a tooltip appears and displays the version description for the version.
- If you are using the mobile version of DataTransfer, select **Settings > Version**, and then select the version you want to use.

Specifying Preferences

Perform one of the following actions.

- Click **Preferences > Set Default Role** to set the current user's role as the default role. When you log in to Perceptive DataTransfer, you are logged in using this user role.
- Click **Preferences > Set Default Version** to set the workspace version of the current user as the default version. When you log in to Perceptive DataTransfer, you are logged in using this workspace version.
- Click **Preferences > Record Grid Dimensions** to specify the number of rows and columns per page to display for records when you open a job.

  The **Record Grid Dimensions** dialog box appears. In the **Columns Per Page** field, type the number of columns to display; in the **Rows Per Page** field, type the number of rows to display. The default value for each field is *20*. The minimum value you can use is *10*, and the maximum is *100*.
- Click **Preferences > Filename in Job Name** to append the filename of the input file to the name of a job when you add jobs. For more information about jobs, refer to Configuring Jobs and Processing Records.
- Click **Display All Match Results** so that when you run jobs in interactive mode, all possible record matches are compiled before the application pauses to allow you to review the possible matches.

  Typically, for jobs running in interactive mode, each match routine in the associated procedures contains a STOP command that pauses job processing, allowing the user to examine the list of possible matches for that match routine. When you set this preference, the STOP command is skipped, and each match routine will run, adding matches to the list of possible matches. If a match routine finds a possible match that is already in the match list, the routine skips the match; the final match list does not contain any duplicates.

In addition, the match results also have an associated rank that is displayed in the results panel, in the # column. The rank is set by the STORE command in each match routine and indicates which match routine found the match. The result list is ordered by ranking—the order in which the match routine discovers the matches.

- Click **Verify Before Running Job** to confirm that you want to run a job. When this option is enabled, when you click Run, the Confirm Run dialog box appears and prompts you to confirm that you want to run the job.
- Click **Skip Previous Results** so that the same possible match record is not displayed twice. For example, when you start processing a record, you receive a list of possible matches, one of which is *John Doe*. If none of the matches are correct, you click the Get More Results button and receive the next possible set of matches. However, if the next set of matches contains *John Doe* again, Perceptive DataTransfer does not display John Doe as a possible match.

Understanding User Roles and Privileges

Your user role and privileges determine what you can do with Perceptive DataTransfer.

The workspace version in which you are working and whether workspaces are shared also determine what you can do. For example, you cannot configure objects, such as data exports, if you are working in a workspace version in staged or active mode.

For more information about configuring roles and privileges, refer to Configuring Users and Privileges.

Administrator privileges determine if you can perform the following actions.

- Add, configure, and delete users.
- Configure system properties.
- Add, configure, and delete share groups.
- View server logs.
- Add, modify, and delete archive rules.
- View workspace details.
- Modify workspace details.
- Use workspace version control.
  - Create new versions in edit mode.
  - Move versions to test mode
  - Move versions to staged mode.
  - Move versions to active mode.
- Work with objects within the workspace.
- Add shared objects.
- Modify shared objects.
- Add, modify, and delete file sources.
- Add, modify, and delete connection groups.
- Add, modify, and delete connections.

- Add, modify, and delete headers.
- Add, modify, and delete verification forms.
- Add, modify, and delete ImageNow index maps.
- Add, modify, and delete data providers.
- Add, modify, and delete document templates.
- Add, modify, and delete procedures.
    - Configure Match logic.
    - Configure Review logic.
    - Configure Upload logic.
    - Configure Match Display logic.
    - Configure Population Selection logic.

User privileges determine if you can perform the following actions.

- Configure the lookup tool.
- Configure the check name tool.
- View data exports.
- Modify data exports.
- Run data exports.
- Export lookups, check names, data exports, log messages, and reports.
- Import workspace versions.
- Run reports.
- Modify reports.
- Modify scheduled reports.
- View report output.
- Delete report output.
- Work within a workspace version.
    - Work within a version in edit mode.
    - Work within a version in test mode.
    - Work within a version in staged mode.
    - Work within a version in active mode.
    - Work within a version in inactive mode.
- Configure jobs.
    - Create new and recurring jobs.
    - Modify recurring jobs.

  - Enable logging of job messages.
  - Enable job hold status.
  - View archive rules for jobs.
  - Create new records.
  - Create matched records.
  - Suspend records.
- Create notification groups.
- Modify notification groups.
- Work within a specific workspace and with specific workspace objects.
- Configure file sources.

Overview of Perceptive DataTransfer Administrative Tasks

When you configure and administer Perceptive DataTransfer, you generally perform the following steps in this order.

1. Define and create the structure of your institution and give privileges for administrators and users to access the appropriate Perceptive DataTransfer features.
2. Configure a workspace in which to work. Workspaces can be private or can be used to share objects, such as headers and procedures, to reuse them across multiple configurations.
3. Configure the logic that Perceptive DataTransfer uses to determine if there is a potential match between a database record and the input data and to determine how information is added or uploaded.
4. Determine the type of input data that you will be loading. For example, if you are loading SAT scores contained in a CSV file, you are importing a delimited file.
5. Create the header. The header defines the input source. For example, if you are importing a delimited file, you configure the header to use the delimited file format for the input file.
6. Configure header variables. These variables define the fields to be loaded from the input file; you determine the names and positions of the fields and associate them with a header variable.
7. Create the verification form, which is used to compare an existing record of information with potentially matching information from the input file, side-by-side.
8. Configure jobs, which consist of all information required to process records.

Viewing Server Logs

You can view a list of server log messages, view the contents of each message, and download messages in either.zip or native format.

1. Select **Administration > Server Logs**.

   A dialog box appears and displays the following columns of information.

   - File Name: Name of the server log file.
   - Size: Size, in Kb, of the log file.
   - Last Modified: Date and time on which the file was last modified.

2. To view the contents of a file, select the file you want to view. To select multiple files:
   - Click a file and press and hold the **Ctrl** key as you click each file with your mouse.
   - Click a file and press and hold the **Shift** key as you click another file to select contiguous files.

   Either right-click the file you want to view and select **View** or select the file and click  . The contents of each file opens in a separate tab.

3. To download the file in ZIP format, select the file you want to download. To select multiple files:
   - Click a file and press and hold the **Ctrl** key as you click each file with your mouse.
   - Click a file and press and hold the **Shift** key as you click another file to select contiguous files.

   Either right-click the file and select **Download as Zip** or select the file and click  ; then, select the location where you want to save the file. When you download multiple files, one ZIP file is created that contains all the log files you selected.

4. To download the file in the application format in which it was authored, select the file you want to download. To select multiple files:
   - Click a file and press and hold the **Ctrl** key as you click each file with your mouse.
   - Click a file and press and hold the **Shift** key as you click another file to select contiguous files.

   Either right-click the file and select **Download Native** or select the file and click ; then, select the location where you want to save the file.

5. To refresh the table of log files, either right-click the table and select **Refresh** or click  .

Configuring PDF to TIFF Properties

You can configure the properties to use when you use the ImageNowAddDoc or NWAddDoc functions to convert a PDF file to a TIFF file. For more information about these functions, refer to Understanding the ImageNowAddDoc function and Understanding the NWAddDoc function.

To configure properties, complete the following steps.

1. Select **Administration > PDF to TIFF**.

   The Configure PDF to TIFF Properties dialog box appears.

2. In the TIFF DPI field, type the DPI value. Values range from 72 to 720; the default value is 240.

3. From the **TIFF Color** drop-down box, select the color properties. Options are: Full Color, Gray Scale, and Black & White. The default value is *Full Color*.
4. From the **Compression Type** drop-down box, select the compression type. Options are: CCITT_RLE, CCITT_T_4, CCITT_T_6, PACKBITS, JPEG, and LZW. The default value is JPEG.
5. You can click **Reset** to reset all values in the dialog box to use the default values.
6. Click **OK** to save your changes and close the dialog box.

Configuring Users and Privileges

When you configure Perceptive DataTransfer users, you define the structure of the institution and configure privileges for users to enable or disable the features they can use.

Configuring Institutions

Refer to the following topics for more information.

- Configuring Organizations
- Configuring Divisions
- Configuring Departments
- Configuring Roles
- Configuring Users

Configuring Organizations

Refer to these topics for more information.

- Adding Organizations
- Renaming Organizations
- Deleting Organizations

Adding Organizations

To add an organization, complete the following steps.

1. Select **Administration > User**.
2. In the **User Administration** dialog box, in the Navigation pane, perform one of the following actions.
   - In desktop view, right-click **Enterprise Hierarchy** and select **Add Organization** from the popup menu.
   - In tablet view, click  .
3. In the Add Organization dialog box, type the name of the organization.
4. Click **OK**. The organization is created and appears in Navigation > Enterprise Hierarchy.
5. Add a division under the organization. Refer to Adding Divisions for more information.

Renaming Organizations

To rename an organization, complete the following steps.

1. Select **Administration > User**.
2. In the **User Administration** dialog box, in the **Navigation** pane, perform one of the following actions.
   - In desktop view, right-click the organization and select **Rename *Organization_Name*** from the popup menu.
   - In tablet view, select the organization and click  .
3. In the **Rename Node** dialog box, type the name of the organization.
4. Click **OK**.

The organization is renamed.

Deleting Organizations

Note You can delete only the lowest-level node in the hierarchy tree. For example, if you want to delete a department that has roles and users configured for it, you must first delete all users, and next all roles, before you can delete the department.

To delete an organization, complete the following steps.

1. Select **Administration > User**.
2. In **the User Administration** dialog box, in the **Navigation** pane, perform one of the following actions. .
   - In desktop view, right-click the organization and select **Delete *Organization_Name*** from the popup menu.
   - In tablet view, select the organization and click  .
3. In the dialog box, click **Yes** to delete the organization.

The organization is deleted, and it is removed from the Navigation pane.

Configuring Divisions

Refer to these topics for more information:

- Adding Divisions
- Renaming Divisions
- Deleting Divisions

Adding Divisions

To add a division, complete the following steps.

1. Select **Adminsitration > User**.
2. In the **User Administration** dialog box, in the **Navigation** pane, perform one of the following actions.
   - In the desktop view, right-click the organization for which you want to add a division and select **Add Division** from the popup menu.

- In the tablet view, select the organization for which you want to add a division click  .

3. In the **Add Division** dialog box, type the name of the division (for example, *Finance*).
4. Click **OK**. The division is created and appears in **Navigation > Enterprise Hierarchy**.
5. Add a department under the division. Refer to Adding Departments for more information.

Renaming Divisions

To rename a division, complete the following steps.

1. Select **Administration > User**.
2. In the **User Administration** dialog box, in the **Navigation** pane, perform one of the following actions to display the **Rename Node** dialog box.
   - In desktop view, right-click the division and select **Rename *Division_Name*** from the popup menu.
   - In tablet view, select the division and click  .
3. Type the name of the division.
4. Click **OK**.

The division is renamed.

Deleting Divisions

Note You can delete only the lowest-level node in the hierarchy tree. For example, if you want to delete a department that has roles and users configured for it, you must first delete all users, and next all roles, before you can delete the department.

To delete a division, complete the following steps.

1. Select **Administration > User**.
2. In the **User Administration** dialog box, in the **Navigation** pane, perform one of the following actions.
   - In desktop view, right-click the division and select **Delete *Division_Name*** from the popup menu.
   - In tablet view, click  .
3. In the dialog box, click **Yes** to delete the division.

The division is deleted, and it is removed from the Navigation pane.

Configuring Departments

Refer to these topics for more information:

- Adding Departments
- Renaming Departments
- Deleting Departments

Adding Departments

To add a department, complete the following steps.

1. Select **Administration > User**.
2. In the **User Administration** dialog box, in the **Navigation** pane, perform one of the following actions.
   - In desktop view, right-click the division for which you want to add a department and select **Add Department** from the popup menu.
   - In tablet view, select the division for which you want to add a department and click  . The Add Department dialog box appears.
3. Type the name of the department (for example, *Accounts Payable*).
4. Click **OK** to create the department, which appears in the **Navigation** pane.
5. You can configure department privileges, which specify the tasks that the department can perform. When you configure privileges at the department level, these privileges are also applied to the users and roles that belong to the department. For more information, refer to Configuring Privileges.
6. Add a role under the department. Refer to Adding Roles for more information.

Renaming Departments

To rename a department, complete the following steps.

1. Select **Administration > User**.
2. In the **User Administration** dialog box, in the **Navigation** pane, perform one of the following actions.
   - In desktop view, right-click the department and select **Rename *Department_Name*** from the popup menu.
   - In tablet view, select the department and click  .
3. In the **Rename Node** dialog box, type the name of the department.
4. Click **OK**.

The department is renamed.

Deleting Departments

Note You can delete only the lowest-level node in the hierarchy tree. For example, if you want to delete a department that has roles and users configured for it, you must first delete all users, and next all roles, before you can delete the department.

To delete a department, complete the following steps.

1. Select **Administration > User**.
2. In the **User Administration** dialog box, in the **Navigation** pane, perform one of the following actions.
   - In desktop view, right-click the department and select **Delete *Department_Name*** from the popup menu.
   - In tablet view, select the department and click  .
3. In the dialog box, click **Yes** to delete the department.

The department is deleted, and it is removed from the Navigation pane.

Configuring Roles

Refer to these topics for more information.

- Adding Roles
- Renaming Roles.
- Deleting Roles

Adding Roles

To add a role, complete the following steps.

1. Select **Administration > User**.
2. In the **User Administrator** dialog box, in the **Navigation** pane, perform one of the following actions.
   - In desktop view, right-click the department for which you want to add a role and select **Add Department Role** from the popup menu.
   - In tablet view, select the department for which you want to add a role and click  .
3. Type the name of the role (for example, *Clerk*).
4. Click **OK**. The role is created and appears in **Navigation > Enterprise Hierarchy**.
5. You can configure role privileges, which specify the tasks that the role can perform. When you configure privileges at the role level, these privileges are also applied to the users that belong to the role. For more information, refer to Configuring Privileges.

   Note You can assign permissions for a workspace and workspace permissions at the role level only.
6. Add a user under the role. Refer to Adding Users for more information.

Renaming Roles

To rename a role, complete the following steps.

1. Select **Administration > User**.
2. In the **User Administration** dialog box, in the **Navigation** pane, perform one of the following actions.
   - In desktop view, right-click the role and select **Rename *Role_Name*** from the popup menu.
   - In tablet view, select the role and click  .
3. In the **Rename Node** dialog box, type the name of the role.
4. Click **OK**.

The role is renamed.

Deleting Roles

Note You can delete only the lowest-level node in the hierarchy tree. For example, if you want to delete a department that has roles and users configured for it, you must first delete all users, and next all roles, before you can delete the department.

To delete a role, complete the following steps.

1. Select **Administration > User**.
2. In the **User Administration** dialog box, in the **Navigation** pane, perform one of the following actions.
   - In desktop view, right-click the role and select **Delete *Role_Name*** from the popup menu.
   - In tablet view, select the role and click  .
3. In the dialog box, to delete the role, click **Yes**.

The role is deleted and removed from Navigation > Enterprise Hierarchy.

Configuring Users

Refer to these topics for more information.

- Adding Users
- Setting Default Roles
- Deleting Users

Adding Users

To add a user, complete the following steps.

1. Select **Administration > User**.
2. In the **User Administration** dialog box, in the **Navigation** pane, perform one of the following actions.
   - In desktop view, right-click the role for which you want to add a user and select **Add User** from the popup menu.
   - In tablet view, select the role for which you want to add a user and click  . The Add User dialog box appears.
3. In the **Add User** dialog box, type the name of the user.
4. Click **OK**. The user is added and appears in the **Navigation > Enterprise Hierarchy**.
5. You can configure user privileges, which specify the tasks that the user can perform. For more information, refer to Configuring Privileges.

Setting Default Roles

You can set a user's role as the default role, if it is not already configured as the default role. Users are logged in to Perceptive DataTransfer with their default role.

1. Select **Administration > User**.
2. In the **User Administrator** dialog box, in the **Navigation** pane, navigate to the user for which you want to set the default role and ;perform one of the following actions.
   - In desktop view, right-click the role and select **Set Default** from the popup menu
   - In tablet view, select the role and click  .

The user role is set as the default.

Deleting Users

To delete a user, complete the following steps.

1. Select **Administration > User**.
2. In the **User Administration** dialog box, in the **Navigation** pane, perform one of the following actions.
   - In desktop view, right-click the user and select **Delete *User_Name*** from the popup menu.
   - In tablet view, select the user and click  .

The user is deleted and removed from Navigation > Enterprise Hierarchy.

Configuring Privileges

You can configure privileges for a department, role, or user. When you configure privileges for a department, those privileges apply to the department and all the roles and users in the department. When you configure privileges for a role, those privileges apply to the role and to all users who belong to the role.

Note Privileges to use a workspace are configured at the role level.

1. Select **Administration > User**.
2. In the **User Administration** dialog box, in the **Navigation** pane, click the department, role, or user for which you want to configure privileges.
3. In the **Department Editor**, **Role Editor**, or **User Editor** pane, configure the department privileges (if you selected a user, in the **User Editor** pane, click the **Permissions** tab). The following table provides descriptions of the privileges.

Note: Click the arrow icon next to each privilege type to expand it and view the privileges. You can check the privilege type check box to select all privileges, or you can check the check box for each individual privilege to select it.

Privilege	Description
Administrative Privileges	
User Administration	Administrator can modify user privileges and can configure email addresses for users to which to send notifications; administrators can modify users and roles within that administrator's department only.
Manage Share Groups	Administrator can add, modify, and delete share groups.
Access Server Logs	Administrator can view server logs.
Configure System Properties	Administrator can configure system properties.
Archive Administration	Administrator can create archive rules for jobs.
Workspace	Contains the following options. • View Details: Administrator can view details about workspace versions and can configure notifications for versions. • Edit Details: Administrator can modify workspace details and can configure notifications for versions.
Version Control	Contains the following options. • Create/Move to Edit: Administrator can create new workspace versions and move workspace versions to edit mode. • Move to Test: Administrator can move workspace versions to test mode. • Move to Staged: Administrator can move workspace versions to staged mode. • Move to Active: Administrator can move workspace versions to active mode.

Privilege	Description
Workspace Configuration	Contains the following options. • Add Shared Objects: Administrator can add shared objects to the workspace. • Edit Shared Objects: Administrator can modify workspace shared objects. • Edit File Sources: Administrators can modify file sources. • Edit Connection Groups: Administrator can modify connection groups. • Edit Connections: Administrator can modify connections. • Edit Headers: Administrator can modify header files. • Edit Verification Forms: Administrator can modify verification forms. • Edit ImageNow Index Map: Administrator can modify ImageNow index maps. • Edit Data Providers: Administrator can configure web service data providers. • Edit Document Templates: Administrators can configure document templates. • Procedures ◦ Edit Match: Administrator can configure match logic. ◦ Edit Review: Administrator can configure review logic. ◦ Edit Upload: Administrator can configure upload logic. ◦ Edit Match Display: Administrator can configure match display logic. ◦ Edit Population Selection: Administrator can configure population selection logic.

Privilege	Description
Tools	Contains the following options. • Lookup: User can configure the lookup tool. • Check Name: User can configure the check name tool. • Export: User can export Perceptive DataTransfer data (for example, lookup values, data provider maps, workspaces, and so on). • Data Exports ◦ Edit Data Exports: User can modify data exports. ◦ Run Data Exports: User can run data exports. ◦ Edit Scheduled Data Exports: Users can add, modify, and delete scheduled data exports. ◦ View Data Export Output: Users can view the output of past data exports. ◦ Delete Data Export Output: Users can delete the output of past data exports. • Reports ◦ Run Reports: User can run reports. ◦ Edit Reports: User can modify reports and can configure notifications for reports. ◦ Edit Scheduled Reports: User can create, modify, and delete scheduled report rules. ◦ View Report Output: User can view output files for reports that have been run. ◦ Delete Report Output: User can delete output files for reports that have been run.
Version	Contains the following options. • Import: User can import workspace versions. • Edit: User can work in workspace versions in edit mode. • Inactive: User can work in workspace versions in inactive mode. • Staged: User can work in workspace versions in staged mode. • Test: User can work in workspace versions in test mode.
Job	Contains the following options. • Create Jobs: User can create jobs and can configure notifications for jobs. • Edit Recurring Jobs: User can modify recurring jobs. • Edit Job Hold Status: User can place a hold on a job to prevent it from being run. • Enable Auditing: User can enable auditing. • View Archived Jobs: User can view archived jobs.

Privilege	Description
Record	Contains the following options. • Create New Record: User can create new records. • Create Match Record: User can set records as matched records. • Create Suspend Record: User can suspend records.
Notifications	Contains the following options. • Create Notification Groups: User can create new notification groups. • Edit Notification Groups: User can modify notification groups.
Workspaces	Workspace privileges are configured at the role level and are applied to all users in the role. A user can use only one workspace at a time. Check the workspace that the user can use, then set the following options. • Headers: Headers contained in the workspace that the user can use. Check the Headers check box to select all headers or check individual headers to select them. • Connection groups: Connection groups contained in the workspace that the user can use. Check the Connection Groups check box to select all connection groups or check individual connection groups to select them. • File Sources: File sources contained in the workspace that the user can use. Check the File Sources check box to select all file sources or check individual file sources to select them.

4. Click the **Settings** tab to configure email addresses for users to which system notifications are sent. For more information, refer to Configuring Notifications.
5. Click **Save** to save your changes.

Configuring Workspaces

To open the Workspace Administration dialog box, select **Administration > Workspace**. The dialog box provides the following information:

- Name. Name of the workspace.
- Description. Brief description of the workspace.
- Sharing. Indicates whether the workspace is private or with a share group.
- Active Version. Version of the workspace that is active.
- Date Created. Date on which the workspace was created.
- Created By. Name of the user who created the workspace.

Refer to the following topics for more information.

- Understanding Workspaces
- Understanding Workspace Sharing
- Understanding Version Control
- Configuring Workspaces
- Using Version Control
- Configuring Share Groups

Understanding Workspaces

Workspaces consist of the objects required for a particular Perceptive DataTransfer configuration.

- Connection groups: A workspace can contain one or more connection groups; a connection group can consist of one or more connections.
- Procedures (code libraries): A workspace contains a set of code libraries, comprising the Perceptive DataTransfer match, review, and upload logic.
- Verification form: A workspace contains a single verification form to be used for all jobs that are run within that workspace.
- Headers: A workspace contains one or more headers, which are used to define input sources. When a job is run within a workspace, one of the headers within the workspace must be selected to use for the job.
- Data exports: A workspace can contain one or more data exports, if they are configured in that workspace. Workspaces do not have to contain data exports.
- File sources: A workspace can contain one or more file sources (both SFTP and server path file locations), which are SFTP sites from which jobs can be downloaded. Workspaces do not have to contain file sources.
- ImageNow index maps: A workspace can optionally contain one or more index maps.
- Variable maps: A workspace can optionally contain one or more variable maps.
- Document templates: A workspace can optionally contain one or more document templates.
- EDI formats: A workspace can optionally contain one or more EDI formats.

You can load multiple configurations, represented by workspaces, into a single Perceptive DataTransfer instance; however, users can only work with one workspace at a time. When you log into Perceptive DataTransfer, you use a default workspace and workspace version but can switch to another workspace version.

You can have multiple versions of a workspace, with each version representing a changeset of the workspace.

Objects can be shared among workspaces. Refer to Understanding Workspace Sharing.

Understanding Workspace Sharing

The following objects can be shared.

- Procedures
- Headers
- Verification forms
- Data exports
- Connection groups
- File sources
- ImageNow index maps
- Document templates
- Variable maps
- EDI formats

You can share an object if multiple workspaces need to use a common object. With sharing, you can reduce the need for duplicating common objects and code, and you can make a single edit and apply it to one object in all workspaces in which it is shared.

For example, you can create a single procedure and share it among multiple workspaces without needing to create duplicate procedures for each workspace.

A workspace can be shared or private.

- Private: The workspace is not shared; therefore, no workspace objects are shared.
- Shared: The workspace can share objects with other workspaces.

The sharing method can be changed at any time; however, you cannot change a shared workspace if shared objects are used by other workspaces and would no longer be shared.

When you share workspace objects, you configure share groups and select the group with which you want to share objects. Any workspaces using the same share group share their objects with other workspaces in this group. Refer to Configuring Share Groups for more information.

Workspaces consist of versions, and objects can be shared depending on the status of your workspace version. Refer to Understanding Object Version Flow for more information.

Understanding Version Control

You can have multiple versions of a workspace; each version represents a changeset of the workspace. There is always at least one version of a workspace.

Refer to Understanding Workspace Version Flow for information on how workspace versions are created and moved.

Understanding Workspace Version Flow

Each workspace version has one of the following statuses.

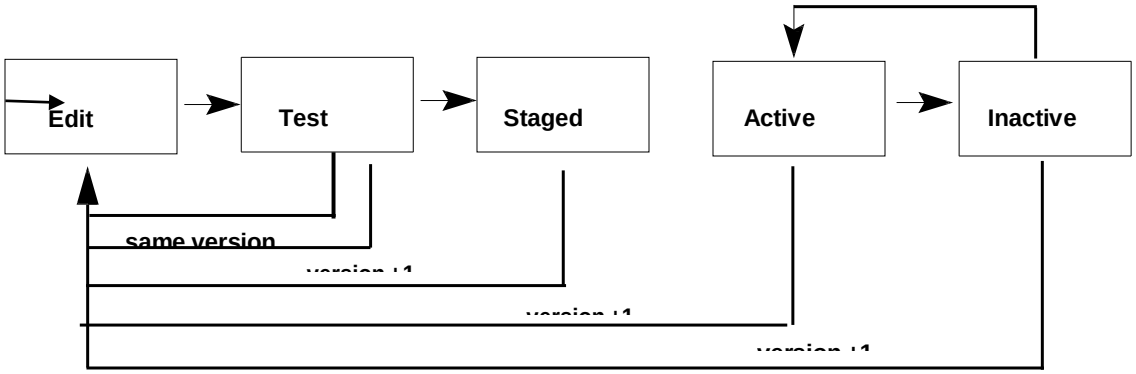
- Edit: This workspace version is a new version. Only one version of the workspace can be in edit status at a time, and this version can be changed by only one user at a time.

Also, only one user at a time can log in to the edit version of a workspace. If a user attempts to log in to the edit workspace version while another user is logged into it and therefore has locked the version, the user receives an appropriate message. In addition, any part of the user interface related to workspaces or versioning is disabled until the user changes to a different role or workspace version.

- Test: This workspace version can be tested by any number of permitted users.
- Staged: This workspace version can be used to view how the workspace will behave in a production environment.
- Active: This workspace version is the current, active version of the workspace. Only one version of the workspace can be active at a time.
- Inactive: This workspace is an older version of the workspace and is no longer in use.

Workspace objects are also versioned and have their own version flow within workspace versions. Refer to Understanding Object Version Flow for more information.

Figure 1: Workspace Version Status Version Flow



Edit

- Objects can be added and modified in edit mode.
- A version in edit mode can be moved to test mode.
- A version in edit mode can be created from a version in staged, active, or inactive mode.
- A version can be reverted to edit mode from test mode.
- A new version in edit mode can be created from a version in test mode.

Test

- A version can enter test mode from edit mode.
- A version in test mode can be moved to edit mode.
- A version in test mode can be moved to staged mode.
- A version in test mode can be deleted.
- A version can be in both edit mode and test mode at the same time; however, there can be only one version each in edit and test mode.

Staged

- A version can enter staged mode from test mode.
- A version in staged mode can move to active mode.
- A version in staged mode can be used to create a version in edit mode.
- A version in staged mode cannot be deleted.

Active

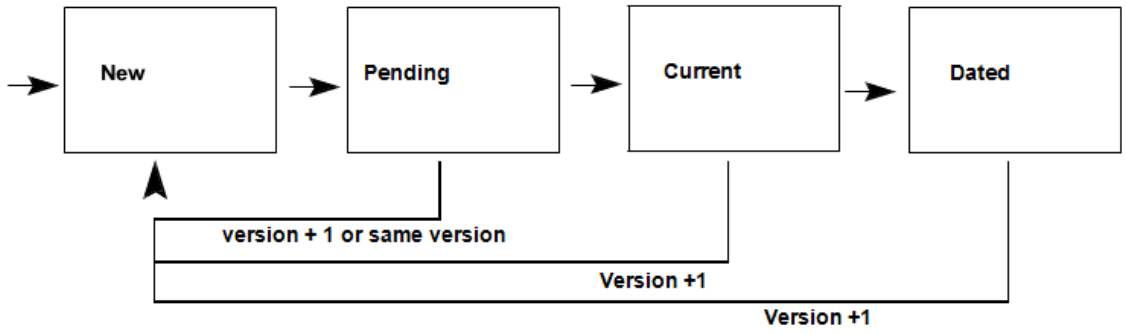
- A version can enter active mode from staged mode.
- A version can enter active mode as a new version created from an inactive version.
- A version in active mode can move to inactive mode (when another version is made active).
- A version in active mode can be used to create a new version in edit mode.
- A version in active mode cannot be deleted.
- Once there is an active version of a workspace, an active version always exists for the workspace.

Inactive

- A version can enter inactive mode from active mode (when another version is made active).
- A version in inactive mode can be used to create a new version in active mode.
- A version in inactive mode can be used to create a new version in edit mode.
- A version in inactive mode cannot be deleted.

Understanding Object Version Flow

Within a workspace version, an object can have a new, pending, current, or dated status.



New

- A new version can be newly created (or copied from an existing object).
- A new version can be created from a pending version.
- A new version can be created from a current version.
- A new version can be created from a dated version.
- A new version can be deleted.

- Only one version of an object can be in new mode.
- A version is in new mode when it was created or modified in a workspace version in edit mode.
- A pending version changes to new version when a workspace is moved from test back to edit mode.

Pending

- A version enters pending mode from new mode when a workspace is moved from edit mode to test mode.
- A version in pending mode is reverted to new mode when a workspace is moved from test mode back to edit mode.
- A version in pending mode is changed to current mode when a workspace is moved from test mode to staged mode.
- A version in pending mode can be removed from the workspace.

Current

- A version enters current mode from pending mode when a workspace is moved from test mode to staged mode.
- A version in current mode can be used to create a version in new mode.
- A version in current mode can move to dated mode (when another version is moved to current).
- A version in current mode can be removed from the workspace.
- A version is in current mode when it was created or modified in a workspace version in staged or active mode.

Dated

- A version can enter dated mode from current mode.
- A version in dated mode can be used to create a version in new mode.
- A version in dated mode can be removed from the workspace.

Rolling Back Versions and Obtaining the Latest Versions of Objects

You can roll back a version of an object (use a previous version of an object) or obtain the latest version of an object if changes have been made and committed to an object.

You can also obtain the latest version of an object if the object has been modified since you added it. For example, you added an object from a workspace in active mode. A new version of the workspace was later added, the shared object was modified, and the version was moved to staged mode. If you obtain the latest object version, these object changes are applied to the object in your workspace.

Configuring Workspaces

Refer to the following topics for more information.

- Importing Workspaces
- Adding Workspaces
- Modifying Workspaces
- Deleting Workspaces

Importing Workspaces

You can import Perceptive DataTransfer 7.*x* workspaces, and you can import Perceptive DataTransfer configurations from versions earlier than 7.0.*x*.

Note You cannot import workspaces in tablet view.

Importing Perceptive DataTransfer 7.x Workspaces

Note Only superadministrators can import workspaces. To import a workspace, complete the following steps.

1. Select **Administration > Workspace**.
2. In the **Workspace Administration** dialog box, click  .
3. In the **Configuration Migration** dialog box, click **Import Workspace** and complete the following substeps.
    1. In the **Workspace Name** field, type a name for the workspace.
    2. In the **Workspace Description** field, type descriptive text about the workspace.
    3. In the **File** field, type the path of the file you want to import, or click **Browse**, navigate to the location of the file, and then select it.
    4. In the **Sharing** field, click the appropriate option.
        - Private: Sets the workspace as a private one; it is not shared, and none of its objects are shared.
        - Shared: Sets the workspace as a shared workspace, which can share its objects with either a specific group of workspaces or all workspaces.
4. If you are sharing the workspace, from the **Group** drop-down box, select the share group to use for the workspace.
7. Click **OK**.

Importing Legacy Configurations

You can import legacy config.dll databases from versions of Perceptive DataTransfer earlier than 7.0.*x*. A new Perceptive DataTransfer workspace is created from the migrated config.dll database. Duplicate objects are shared with existing Perceptive DataTransfer workspaces when applicable. Perceptive DataTransfer manages importing of Perceptive DataTransfer functions as follows.

- Enterprise Hierarchy

  Legacy hierarchies are merged in a single hierarchy. A node in the hierarchy is merged with an existing node if the node names match and each subsequent parent in the tree matches. If no matches are found, the node is created as new, either as the beginning of a new tree for an organization or as the child to a matching parent in other cases.

- Database connections

  For each migrated config.dll, a new connection group is created. A default connection, corresponding to the legacy ODBC connection selected at Perceptive DataTransfer log in, is created as the primary connection of the group. All other legacy secondary connections are added to the group. Names and descriptions are migrated, but legacy connection details are omitted. Current connection details are provided manually after migration.

- Code libraries

  Match, Review, and Upload libraries, along with their corresponding procedures and SQL blocks, are migrated.

  Libraries can be merged or not merged depending on migration parameters. If merged, the existing repository is searched for matching libraries. A library is considered a match if its properties are identical, including all its procedure and SQL blocks. If a match is found, and that match is shared in the same share group, then the migrated collection will share a link to the matched library with any workspaces that already share the library. If there is no match, or the matches are not shared, then the library is created as new, along with all its procedures and SQL blocks.

  Procedures and SQL blocks may be merged or not merged depending on migration parameters. If merged, the existing repository is searched for matching procedures. A procedure is considered a match if its properties and text are identical. If a match is found, and that match is shared in the same share group, then the migrated workspace will share a link to the matched procedure with any workspaces that already share the procedure. If there is no match, or the matches are not shared, then the procedure is created as new. The same logic applies to SQL blocks.

  Libraries, procedures, and SQL blocks are migrated as is. Legacy conditions within their properties and text are automatically resolved after the config.dll migration, but some manual corrections may need to be made if there are issues with legacy code.

- Program variables

  Program variables are migrated from the legacy database to the new Perceptive DataTransfer 7.*x* schema as is. Program variables contain links to verification form objects; however, links between program variables and form objects are not maintained. Data typing issues or other legacy concerns are resolved after the config.dll migration. All program variables are created as new and are not shared with other workspaces.

- Headers

  Headers can be merged with existing headers or created as new, depending on migration parameters. If merged, the existing repository is searched for matching headers. A header is considered a match if all its properties, including its type and subsequent details, are identical. If a match is found, and that match is shared with the same share group, then the migrated workspace will share a link to the matched header with any workspaces that already share the header. If there is no match, or the matches are not shared, then the header is created as new.

- Data exports

  Data exports can be merged with existing data exports or created as new, depending on migration parameters. If merged, the existing repository is searched for matching data exports. A data export is considered a match if all its properties are identical. If a match is found, and that match is shared with the same share group, then the migrated workspace will share a link to the matched data export with any workspaces that already share the data export. If there is no match, or the matches are not shared, then the data export is created as new.

Notes

- Only superadministrators can import legacy configurations.
- *BEGIN TRAN* in SQL statements are removed from code during migration.

To import a legacy configuration, complete the following steps.

1. Select **Administration > Workspace**.

2. In the **Workspace Administration** dialog box, click  .
3. In the **Configuration Migration** dialog box, click **Import Legacy Configuration** and complete the following substeps.
    1. In the **Workspace Name** field, type a name for the workspace.
    2. In the **Workspace Description** field, type descriptive text about the workspace.
    3. In the **Config.dll Upload** field, type the path of the file you want to import, or click **Browse**, navigate to the location of the file, and then select it.
    4. In the **Sharing** field, click the appropriate option.
        - Private: Sets the workspace as a private one; it is not shared, and none of its objects are shared.
        - Shared: Sets the workspace as a shared workspace, which can share its objects with workspaces in the same share group.
4. For a shared workspace, from the **Group** drop-down box, select the share group to use for the workspace. If you are sharing a workspace, you must select the appropriate share group.
5. Check the **Merge Headers** check box to merge legacy headers with Perceptive DataTransfer 7.*x* headers.
6. Check the **Merge Data Exports** check box to merge legacy data exports with Perceptive DataTransfer 7.*x* data exports.
7. Check the **Merge NQL Procedures** check box to merge procedures.
8. Click **Migrate**.

Adding Workspaces

Note: Only superadministrators can add new workspaces.

To add a new workspace, complete the following steps.

1. Select **Administration > Workspace**.
2. In the **Workspace Administration** dialog box, click  .
3. In the **Add Workspace** dialog box, complete the following substeps.
    1. In the **Name** field, type the name of the workspace.
    2. In the **Description** field, type the description of the workspace.
    3. In the **Sharing** field, click the appropriate option.
4. If you are sharing a workspace, from the **Group** drop-down box, select the share group to use.
5. Click **OK**. The workspace is added and displays in the Workspace Manager dialog box.
6. To use this workspace, give yourself the appropriate privileges. Refer to Configuring Users and Privileges.

Modifying Workspaces

To modify a workspace, complete the following steps.

1. Select **Administration > Workspace**.
2. In the **Workspace Administration** dialog box, select the workspace you want to modify, and then either right-click it and select **Edit** or click  .
3. In the **Workspace Details** dialog box, complete the following substeps.
    1. In the **Name** field, type the name of the workspace.
    2. In the **Description** field, type the description of the workspace.
    3. In the **Sharing** field, click the appropriate option.
        - Private. Sets the workspace as a private one, if it is not shared, and none of its objects are shared.
        - Shared. Sets the workspace as a standard workspace, which can share its objects with workspaces in the same share group.
4. If you are sharing the workspace, from the **Group** drop-down box, select the share group to us.
5. Click **Save**.

The workspace is modified, and its updated information displays in the Workspace Manager dialog box.

Deleting Workspaces

Note: Only superadministrators can delete workspaces.

To delete a workspace, complete the following steps.

1. Select **Administration > Workspaces**.
2. In the **Workspace Manager** dialog box, select the workspace you want to delete, and then either right-click it and select **Remove,** or click  .

The workspace is deleted and is removed from the Workspace Manager dialog box.

Using Version Control

Refer to the following topics for more information.

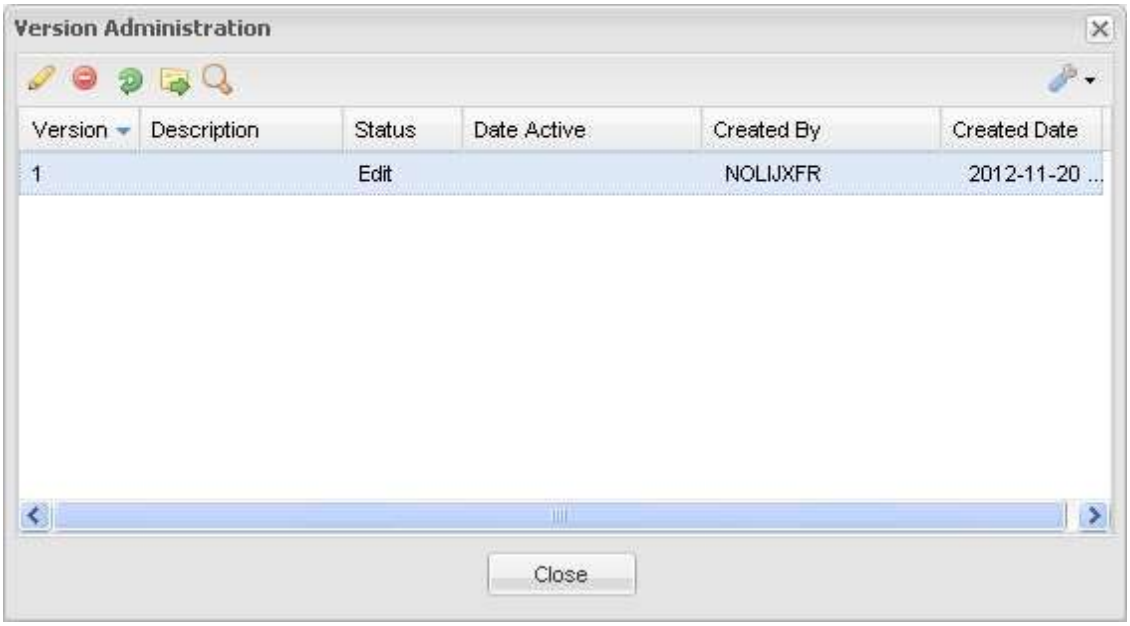
- Modifying Versions
- Viewing Version Details
- Moving Versions to Test Mode
- Reverting a Version to Edit Mode
- Moving Versions to Staged Mode
- Adding Versions
- Moving Versions to Active
- Obtaining the Latest Versions of Shared Objects
- Importing Versions

- Exporting Versions
- Configuring Workspace Version Notifications

The Version Administration dialog box displays when you select a workspace in the Workspace Manger dialog box and either right-click it and select Versions, or click . It displays the following information.

- Version: Numerical value of the version.
- Description: Description of the version.
- Status: Status of the version.
- Date Active: Date on which the version was activated. If the version is not active, this column displays a blank row of information.
- Created By: Name of the user who created the version.
- Create Date: Date on which the version was created.

Figure 3: Version Administration Dialog Box



Modifying Versions

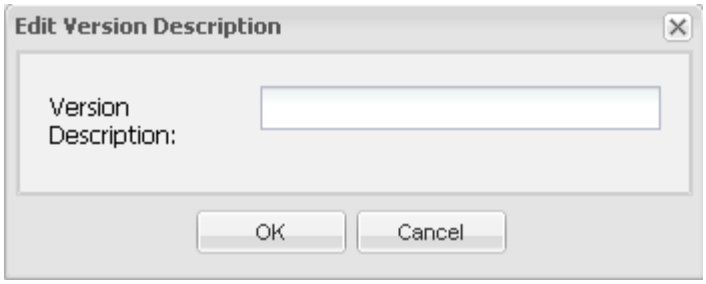
Note: You can modify only the version in which you are working if it is in edit or test mode.

To modify a version name, complete the following steps.

1. Select **Administration > Workspace**.
2. In the **Workspace Manager** dialog box, select the workspace you want to modify and either right-click it and select **Versions**, or click .

3. In the **Version Administration** dialog box, select the version you want to modify and either right-click it and select **Edit Description**, or click .
4. In the **Edit Version Description** dialog box, type descriptive text about the version and click **OK**.

   The version description is updated and displays in the **Version Administration** dialog box.



Viewing Version Details

To view version information, complete the following steps.

1. Select **Administration > Workspace**.
2. In the **Workspace Administration** dialog box, click the workspace for which you want to view version details and then either right-click it and select **Version**, or click .
3. In the **Version Administration** dialog box, either right-click the version and select **View**, or click .

   A dialog box displays for the version and displays the following information.

   - Action: Action that was taken on an object (*Add*, *Modify*, or *Delete*).
   - Type: Type of object that was added, modified or deleted.
   - Name: Name of the object that was added, modified, or deleted.
   - Date: Date on which the action occurred.

- User: Username of the user who added, modified, or deleted the object.
- Comment: Comment that was added to the version entry.

Adding Versions

You can add a new version of a workspace if you do not have any other versions with edit status. The new version you add has a status of edit.

A workstation version in edit mode can be created from a version in staged, active, inactive, or test mode. A version in test mode can be reverted to edit mode if no other edit mode version exist.

To revert a version in a test mode back to edit mode, follow the procedure in Reverting a Version to Edit Mode.

Use the following procedures to create a new workspace version in edit mode from workspace versions in staged, active, inactive, or test mode.

1. Select **Administration > Workspace**.
2. In the **Workspace Administration** dialog box, click the workspace to which you want to add a new version and either right-click it and select **Version**, or click .

3. In the **Version Administration** dialog box, select the workspace version from which you want to create a new version in edit mode.

   When you create a new version from the selected version, a copy of the selected version is created and is placed in edit mode.

4. Click  .
5. In the dialog box, click **Yes** to create a new version.
6. In the **Add Version** dialog box, in the Description field, type descriptive text about the version and click **OK**.

The version is created and displays in the Version Administration dialog box.

Moving Versions to TestMode

Note: You can move a version from only edit mode to test mode. If you move a version from edit mode to test mode, any existing version in test mode become inactive.

To move a version to test mode, complete the following steps.

1. Select **Administration > Workspace**.
2. In the **Workspace Administration** dialog box, click the workspace whose version you want to test and either right-click it and select **Version** or click  .
3. In the **Version Administration** dialog box, select the version you want to advance to testing and either right-click it and select **Test** or click  . A dialog box appears and prompts you to confirm that you want to advance the version to testing.
4. In the dialog box, click **Yes** to change the status to test.

The version is advanced to test mode, and its information is updated in the Version Administration dialog box.

Reverting a Version to Edit Mode

Note: You can revert a version from only test mode to edit mode. You can revert a version from test mode to edit mode only if there are no versions already in edit mode.

To revert a version back to edit mode from test mode, complete the following steps.

1. Select **Administration > Workspace**.
2. In the **Workspace Administration** dialog box, click the workspace whose version you want to revert to editing and either right-click it and select **Version** or click  .
3. In the **Version Administration** dialog box, select the version you want to revert to editing and either right-click it and select **Edit** or click  .
4. In the dialog box, click **Yes** to change the status to edit.

The version is reverted to edit status, and its information is updated in the Version Administration dialog box.

Moving Version to Stage Mode

Note: You can move version to staged mode only from test mode.

1. Select **Administration > Workspace**.
2. In the **Workspace Administration** dialog box, click the workspace whose version you want to advance to staging and either right-click it and select **Version** or click  .
3. In the **Version Administration** dialog box, s appears. select the version you want to advance to staging and either right-click it and select **Stage** or click  .
4. In the dialog box, click **Yes** to change the status to staged.

The version is changed to staged status, and its information is updated in the Version Administration dialog box.

Moving Versions to Active

Note: You can move a version to active with the status of *staged* or *inactive* only.

1. Select **Administration > Workspace**.
2. In the **Workspace Administration** dialog box, click the workspace whose version you want to activate and either right-click it and select **Version,** or click  .
3. In the Version Administration dialog box, select the version you want to activate and either right-click it and select **Activate,** or click  .
4. In the dialog box, click **Yes** to activate the version.

The version is activated, and its information is updated in the Version Administration dialog box.

Obtaining the Latest Versions of Shared Objects

If changes have been made to and committed to shared objects, you can obtain the latest versions of all shared objects.

Note: You can also obtain the latest versions of individual objects (procedures, forms, data exports, headers, and connection groups) in the appropriate dialog boxes or application windows for the objects.

1. Select **Administration > Workspace**.
2. In the **Workspace Administration** dialog box, click the workspace whose latest version you want to obtain and either right-click it and select **Version,** or click  .
3. In the **Version Administration** dialog box, select the version and click  .

The latest objects are obtained and are loaded into the workspace.

Importing Versions

Note: You cannot import versions in tablet view.

To import a version from an XML file, complete the following steps.

1. Select **Administration > Workspace**.
2. In the **Workspace Administration** dialog box, click the workspace for which you want to import a version and either right-click it and select **Version,** or click .
3. In the **Version Administration** dialog box, click and select **Import**.
4. In the **Import Version** dialog box, click **Browse** and navigate to the version you want to import, select it, and click **OK**.

The version is imported and appears in the Version Administration dialog box.

Exporting Versions

Note You cannot export versions in tablet view.

To export a version in an XML file, complete the following steps.

1. Select **Administration > Workspace**.
2. In the **Workspace Administration** dialog box, click the workspace for which you want to export a version and either right-click it and select **Version**, or click .
3. In the **Version Administration** dialog box, select the version you want to export.
4. Click and select **Export**.
5. In the **Workspace Export** dialog box, click the **Full** option if you want to export all objects in the workspace; click **Changes** if you want to only export changes, and click **OK**.
6. Save the file in the appropriate location.

Configuring Workspace Version Notifications

You can configure Perceptive DataTransfer to send email notifications to specified users about when certain workspace version activities occur. When you select **Administration > Workspace** and click , the Version Notification Configuration dialog box appears, from which you can configure notifications. For more information about configuring notifications, refer to Configuring Notifications.

Configuring Share Groups

Workspaces and their objects can be shared among share groups. Create a share group and configure workspaces to use that share group so that objects can be shared.

Adding Share Groups

To add a share group, complete the following steps.

1. Select **Administration > Share Groups**.
2. In the **Share Group Administration** dialog box, click .
3. In the **Add Share Group** dialog box, complete the following substeps.

1. In the **Share Group Name** field, type the name of the share group.
2. In the **Description** field, type descriptive text for the share group.

9. Click **OK**.

The share group is created and displays in the Share Group Administration dialog box.

When you share a workspace, you select the share group to use. Workspaces that use this share group can share their objects with all other workspaces using the share group.

Modifying Share Groups

To modify a share group, complete the following steps.

1. Select **Administration > Share Groups**.
2. In the **Share Group Administration** dialog box, select the share group you want to modify and either right-click it and select **Edit,** or click  .
3. In the **Edit Share Group** dialog box, complete the following substeps.
    1. In the **Share Group Name** field, type the name of the share group.
    2. In the **Description** field, type descriptive text for the share group.
4. Click **OK**.

The share group is modified and updated in the Share Group Administration dialog box.

Deleting Share Groups

Note: You cannot delete a share group if a workspace is using it.

To delete a share group, complete the following steps.

1. Select **Administration > Share Groups**.
2. In the **Share Group Administration** dialog box, select the share group you want to delete and either right-click it and select **Delete,** or click  .
3. In the dialog box, click **Yes** to delete the share group.

The group is deleted and is removed from the Share Group Administration dialog box.

Configuring connection groups and database connectors

Connection groups consist of a group of database connections and are associated with a workspace; connection groups and their connections can be shared among workspaces.

You can import connection groups from and export connection groups to an XML file.

Refer to the following topics for more information.

- Configuring connection groups
- Configuring connections

Configuring connection groups

Refer to the following topics for more information.

- Adding connection groups
- Modifying connection groups
- Deleting connection groups
- Copying connection groups
- Removing sharing on shared connection groups
- Rolling back connection groups
- Exporting connection groups
- Importing connection groups

Adding connection groups

You can add a new connection group or add a connection group that another workspace is sharing by completing the following steps.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, click  .
3. In the **Add Connection Group** dialog box, perform one of the following options.
   - To add a connection group, complete the following substeps.
     1. Click the **Add Connection Group** tab.
     2. In the **Connection Group Name** field, type the name of the connection group.
     3. In the **Description** field, type descriptive text about the connection group.
     4. Click **OK**.
   - To add a connection group that is shared by another workspace, complete the following substeps.
     1. Click the **Add Shared Connection Group** tab.
     2. From the **Workspace** list, select the workspace that is sharing the connection group you want to use.

3. Select the connection group you want to use.
4. Click **OK**.

   The connection group is added and appears in the **Connection Group Administration** dialog box.

   A database connection with the name *Primary* is automatically added to the connection group. A connection group must always have a primary database connection. You can add additional database connections to the group. Refer to Adding and modifying database connections.

Modifying connection groups

To modify a connection group, complete the following steps.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, click the connection group you want to modify and either click  or right-click the connection group and select **Edit Connection Group**.
3. In the **Edit Connection Group** dialog box, in the **Connection Group Name** field, type the name of the connection group.
4. In the **Description** field, type a brief description for the connection group and click **OK**.

The connection group is modified, and its updated information appears in the **Connection Group Administration** dialog box.

Deleting connection groups

To delete a connection group, complete the following steps.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, click the connection group you want to delete and click  .

Copying connection groups

To copy a connection, complete the following steps.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, click the connection group you want to copy and either right-click it and select **Copy Connection Group**, or click  .
3. In the **Rename Copied Connection Group** dialog box, in the **Name** field, type the name of the connection group and click **OK**.

The connection group is copied, and the new connection group you created is added to the **Connection Group Administration** dialog box.

Removing sharing on shared connections groups

To remove sharing on a connection group that is shared with another workspace, complete the following steps.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, select the connection group for which you want to remove sharing and either right-click and select **Remove Sharing**, or click  .

Sharing is removed on the connection group.

Rolling back connection groups

If changes have been made and committed to a connection group, you can use a previous version (*roll back*) of it.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, click the connection group you want to roll back and either right-click it and select Rollback or click  .

   The **Rollback Manager** dialog box appears and displays the following information.

   - **Name**: Name of the connection group.
   - **Status**: Status of the connection group (*current* or *dated*).
   - **Version Date**: Date and time on which the version was created.
   - **Created By**: User name of the user who created the connection group.
   - **Comments**: Comments that were added by the user.
3. Select the previous version to which you want to revert and click **Rollback**. The previous version is loaded.

Obtaining the latest versions of connection groups

If changes have been made to and committed to a connection group, you can obtain the latest version of it by completing the following steps.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, click the connection group for which you want to obtain the latest version and click  .

Exporting connection groups

You can export connection groups to an XML file by completing the following steps.

Note You cannot export connection groups in tablet view.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, select the connection group you want to export.

3. Click  and select **Export**.
4. Save the file to the appropriate location.

Importing connection groups

You can import connection groups from an XML file by completing the following steps.

Note: You cannot import connection groups in tablet view.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, click  and select **Import**.
3. In the **Import Connection Groups** dialog box, click the **Browse** button and navigate to the file that you want to import or type the location of the file in the **File** field.
4. Check the **Update Existing Entities** check box to update an existing connection group, with the same name as the connection group you are importing, with the connection group you are importing.

   If you do not check this check box, and there is an existing connection group with the same name as the connection group you are importing, *DUPLICATE-* is appended to the start of the name of the connection group you are importing; the existing connection group is not updated. If a connection group exists with the name DUPLICATE-<*connection_group_name*>, *DUPLICATE(2)-* is appended to the start of the name. The number continues to be incremented so that all connection groups have unique names.
5. Click **OK**.

Configuring connections

Refer to the following topics for more information.

- Adding and modifying database connections
- Adding and modifying Nolij Web connections
- Adding and modifying Hobsons Connect CRM connections
- Adding and modifying integration server connections
- Adding and modifying web service connections
- Adding and modifying SOAP web service connections
- Deleting connections
- Testing connections

Adding and modifying database connections

To add or modify a database connection, complete the following steps.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, complete one of the following actions.
   - To add a new database connection, complete the following substeps.
     1. Select the connection group to which you want to add a new connection and either right-click it and click .
     2. In the **Add Connection** dialog box, from the **Type** drop-down box, select **Standard**,
   - To modify a database connection, select the connection you want to modify and click .
     1. In **Add Database Connection** or **Edit Database Connection** dialog box, in the **Name** field, type the name of the connection.

        Note: A primary database connection with the name *Primary* is automatically created when you create a connection group. You can have only one primary database per group. You cannot name another database connection *Primary*.
     2. In the **Description** field, type a brief description of the connection.
     3. In the **URL** field, type the URL of the database.
     4. In the **Username** field, type the user name.
     5. In the **Password** field, type the password.
     6. In the **DB Driver** field, type the name of the driver used to connect to the database.
     7. In the **DB Name** field, type the name of the database.
     8. Click **OK**.

Adding and modifying Nolij Web connections

To add or modify a Nolij Web connection, complete the following steps.

Note You can configure only one Nolij Web connection per connection group.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, perform one of the following actions.
   - To add a new Nolij Web connection, complete the following substeps.
     1. Select the connection group to which you want to add a new connection and click .
     2. In the **Add Connection** dialog box, from the **Type** drop-down box, select **Nolij Web.**
   - To modify a Nolij Web connection, select the connection you want to modify and either right-click it and select **Edit Connection** or click .
3. In the **Add Nolij Web Connection** or **Edit Nolij Web Connection** dialog box, in the **Name** field, type the name of the connection.
4. In the **Description** field, type a brief description of the connection.
5. In the **URL** field, type the URL of the Nolij Web instance (for example, http://*localhost*/NolijWeb/).

6. In the **Username** field, type the user name.
7. In the **Password** field, type the password.
8. In the **CAS URL** field, type the URL of the CAS server.
9. Check the **Is CAS** check box to indicate that this connection uses CAS authentication.
10. Click **OK**.

Adding and modifying Hobsons Connect CRM connecting

You can update Hobsons data using Hobsons Connect CRM web services. To do so, you create a connection group that contains a Hobsons Connect CRM connection. You can also create a Hobsons Connect CRM header and add and run a job using the Hobsons CRM Connect connection and header. You can also use the appropriate NQL syntax in the Perceptive DataTransfer logic to execute the Hobsons web service.

Note You can configure only one Hobsons CRM Connect connection per connection group.

To add or modify a Hobsons Connect CRM connection, complete the following steps.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, complete one of the following actions.
   - To add a new Hobsons Connect CRM connection, complete the following substeps.
     1. Select the connection group to which you want to add a new connection and click  .
     2. In the **Add Connection** dialog box, from the **Type** drop-down box, select **Hobsons Connect CRM**.
   - To modify a Hobsons Connect CRM connection, select the connection you want to modify and click  .
3. In **Add Hobsons Connect CRM** or **Edit Hobsons Connect CRM** dialog box, in the **Name** field, type the name of the connection.
4. In the **Description** field, type descriptive text about the connection.
5. In the **Client URL** field, type the URL used by clients to access web services.
6. In the **Action URL** field, type the URL used for SOAP (Simple Object Access Protocol) web services.
7. In the **Client Name** field, type the client name used for the connection.
8. In the **Pass Key** field, type the pass key used for the connection.
9. Click **OK**.

The connection is added and appears in the **Connection Group Administration** dialog box.

You configure jobs to use connection groups; refer to Configuring Jobs and Processing Records. When you configure a job to use a connection group with a Hobsons connection and a Hobsons Connect CRM header, you can also configure the logic used by the job to use the appropriate NQL syntax, as follows.

```
HobsonsUpdate(<web service name>, <contact id>, <attribute name 1>, <attribute value 1>, <attribute name 2>, <attribute value 2>, ... , <attribute name n>, <attribute value n>)
```

The required parameters are:

- <*web service name*>: the name of the Hobsons Connect CRM web service (for example,
- *UpdateContact*)
- <*contact id*>: the ID value of the contact record to update.

The additional parameters are any number of attribute name or value pairs. Both components of the pair are strings; the first string identifies the name of the Hobsons Connect CRM attribute to update, and the second string is the new value for the specified attribute.

The HobsonsUpdate function returns a string. If the update is successful, an empty string is returned. If it is not successful, a corresponding error message is returned.

An example of the syntax is

```
let $err = HobsonsUpdate('UpdateContact', $id, 'first_name', $fname, 'last_name',
$lname, 'dob', $dob')
```

This syntax runs the *UpdateContact* web service on the contact with id *$id* and updates the *first_name* attribute to the value in *$fname*, the *last_name* attribute to the value in *$lname*, and so on.

For more information about procedures, refer to Configuring Procedures. For detailed information about configuring logic, refer to Appendix A: Programming Concepts.

Adding and modifying integration server connections

Configure an Integration Server connection when you want to interact with ImageNow, using Perceptive DataTransfer.

You use Integration Server connections to obtain document information about the ImageNow documents you want to upload. When you configure index maps, you use the connection to obtain the information, which is displayed as Perceptive DataTransfer program variables, and map the ImageNow document properties to the corresponding program variables. Refer to Configuring ImageNow Index Maps.

Use the ImageNowAddDoc function in your code to specify the parameters Perceptive DataTransfer uses to upload ImageNow documents. Refer to Understanding the ImageNowAddDoc function.

Note You can configure only one ImageNow connection per connection group.

To add or modify an Integration Server connection, complete the following actions.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, complete one of the following actions.

To add a new Integration Server connection, complete the following steps.

1. Select the connection group to which you want to add a new connection and click .
2. In the **Add Connection** dialog box, from the **Type** drop-down box, select **Integration Server**.
3. Select the connection group to which you want to add a new connection and click .
4. In the **Add Connection** dialog box, from the **Type** drop-down box, select **Integration Server**.
5. In the **Description** field, type descriptive text about the connection
6. In the **URL** field, type the username used for the connection.
7. In the **Password** field, type the password used for the connection.
8. Click **OK**.

Adding and modifying web service connections

Create a web service connection when you are using web services to retrieve or update data. For more information, refer to Configuring data providers.

Refer to the following topics for more information.

- Adding and modifying web service connections without authentication
- Adding and modifying session header connections
- Adding and modifying session cookie connections
- Adding and modifying HMAC connections

Adding and modifying web service connections without authentication

To add or modify web service connection that does not use an authentication scheme, complete the following steps.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, perform one of the following actions.
   - To add a new connection, complete the following substeps.
     1. Select the connection group to which you want to add a new connection and click .
     2. In the **Add Connection** dialog box, from the **Type** drop-down box, select **Web Service**.
     3. From the **Authentication** drop-down box, select **Not Applicable** and click **OK**.
   - To modify a web service connection, select the connection you want to modify and click .
3. In the **Add Web Service** or **Edit Web Service** dialog box, in the **Name** field, type the name of the web service connection.
4. In the **Description** field, type descriptive text about the web service connection.
5. In the **Client URL** field, provide the beginning portion of the URL that runs the web service. If you have multiple web services that start with the same portion of the URL, you can provide this common portion in the URL field and share the connection among multiple web services.
6. Click **OK**.

Adding and modifying session header connections

A session header connection extracts a session ID or token from the response sent by an initial request to a specified URL. The authentication request is executed with the parameters you specify, and the session ID or token is extracted from the response. For each subsequent web service call using this connection, the extracted session ID token is then passed into the header field you specify.

To add or modify a session header connection, complete the following actions.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, perform one of the following actions.
   - To add a new connection, complete the following substeps.
     1. Select the connection group to which you want to add a new connection and click  .
     2. In the **Add Connection** dialog box, from the **Type** drop-down box, select **Web Service.**
     3. From the **Authentication** drop-down box, select **Session Header** and click **OK**.
   - To modify a web service connection, select the connecting you want to modify and click  .
3. In **Add Web Service** or **Edit Web Service** dialog box, in the **Name** field, type the name of the session header web service connection.
4. In the **Description** field, type descriptive text about the session header web service connection.
5. In the **Client URL** field, provide the beginning portion of the URL that runs the session header web service. If you have multiple web services that start with the same portion of the URL, you can provide this common portion in the URL field and share the connection among multiple web services.
6. In the **Authentication** pane, specify session header authentication options by completing the following substeps.
   4. In the **URL** field, type the URL from which to receive the response.
   5. From the **HTTP Method** drop-down box, select the HTTP method used.
   6. In the **Content Type** field, type the content type to send to the URL. For example:
      - For JSON authentication service, the content type is *application/JSON*.
      - For XML authentication service, the content type is *application/XML*.
   7. In the **Content to Send** field, provide the content (for example, the request payload) to send. For example:
      - For JSON authentication service, the content to be sent is in JSON format.
      - For XML authentication service, the content to be sent is in XML format or is part of a URL string.
   8. In the **Header Field** field, type the header field into which the session ID or token is passed.

7. If your authentication request requires you to include certain header properties in the request, you can optionally configure request headers information. For example, if you are making an authentication request to the Integration Server, which requires navigating to the appropriate URL and including the Integration Server user name and password in the request header, you can add a request header to the authentication request. Complete one of the following actions.
   - To add a new request header, In the **Request Headers** pane, complete the following substeps.
     9. Click  .
     10. In the **Add Request Header** dialog box, in the **Name** field, type the name of the request header. For example, if you are specifying an Integration Server user name, you could type `X-IntegrationServer-Username`.
     11. In the **Value** field, type the value of the request header. For example, for an X-IntegrationServer-Username request header with Administrator as the user name, you would type `Administrator`.
     12. Click **OK**.
   - To modify a request header, complete the following substeps.
     1. Select the request header you want to modify in the **Request Headers** pane and click  .
     2. In the **Edit Request Header** dialog box, in the **Value** field, type the value of the request header.
     3. Click **OK**.
   - To delete a request header, complete the following substeps.
     1. Select the request header you want to modify in **the Request Headers** pane and click  .
     2. In the **Edit Request Header** dialog box, in the **Value** field, type the value of the request header.
     3. Click **OK**.
   - To delete a request header, complete the following substeps.
     1. Select the request header you want to delete and click  .

        A dialog box appears, prompting you to confirm that you want to delete the requested header.
     2. To delete the request header, click **OK**.
8. Click **OK**.

Adding and modifying session cookie connections

A session cookie connection extracts a session ID or token from the cookie sent by an initial request to a specified URL. The authentication request is executed with the parameters you specify, and the session ID or token is extracted from the cookie set by the response. For each subsequent web service call using this connection, the extracted session ID or token is then passed into the cookie you specify.

To add a session cookie web service connection, complete the following steps.

1. Select **Configuration** > **Connections**.
2. In the **Connection Group Administration** dialog box, complete one of the following actions.
   - To add a new session cookie connection, complete the following substeps.
     1. Select the connection group to which you want to add a new connection and click .
     2. In the **Add Connection** dialog box, from the **Type** drop-down box, select **Web Service**.
     3. From the **Authentication** drop-down box, select **Session Cookie** and click **OK**.
   - To modify a session cookie web service connection, select the connection you want to modify and click .
3. In the **Add Web Service Connection** or **Edit Web Service Connection** dialog box, in the Name field, type the name of the session cookie web service connection.
4. In the **Description** field, type descriptive text about the session cookie web service connection.
5. In the Client URL field, provide the beginning portion of the URL that runs the session cookie web service. If you have multiple web services that start with the same portion of the URL, you can provide this common portion in the URL field and share the connection among multiple web services.
6. In the Session Cookie pane, specify session cookie authentication options by completing the following substeps
     1. In the **URL** field, type the URL from which to receive the response.
     2. From the **HTTP Method** drop-down box, select the HTTP method used.
     3. In the **Content Type** field, type the content type to send to the URL. For example,
   - For JSON authentication service, the content type is *application/JSON*.
   - For XML authentication service, the content type is *application/XML*.
     4. In the **Content to Send** field, provide the content (for example, the request payload) to send. For example,
   - For JSON authentication service, the content to be sent is in JSON format.
   - For XML authentication service, the content to be sent is in XML format or is part of a URL string.
     5. In the **Cookie Field**, type the cookie into which the session ID or token is passed.

7. If your authentication request requires you to include certain header properties in the request, you can optionally configure request headers information. For example, if you are making an authentication request to the Integration Server, which requires navigating to the appropriate URL and including the Integration Server user name and password in the request header, you can add a request header to the authentication request. Complete one of the following actions.
   - To add a new request header, in the **Request Headers** pane, complete the following substeps.
     1. Click  .
     2. In the **Add Request Header** dialog box, in the **Name** field, type the name of the request header. For example, if you are specifying an Integration Server user name, you could type `X- IntegrationServer-Username`.
     3. In the **Value** field, type the value of the request header. For example, for an X-IntegrationServer-Username request header with Administrator as the user name, you would type `Administrator`.
     4. Click **OK**.
   - To modify a request header, complete the following substeps.
     1. Select the request header you want to modify in the **Request Headers** pane and click  .
     2. In the **Edit Request Header** dialog box, in the **Value** field, type the value of the request header.
     3. Click **OK**.
   - To delete a request header, complete the following substeps.
     1. Select the request header you want to delete and click  .

        A dialog box appears, prompting you to confirm that you want to delete the request header.
     2. To delete the request header, click **OK**.
8. Click **OK**.

Adding and modifying HMAC connections

An HMAC (hash-based method authentication code) web service connection creates a customizable HMAC signature included in an authorization header for all web service requests using that connection.

To add or modify an HMAC web service connection, complete the following steps.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, complete one of the following actions.
   - To add a new HMAC web service connection, complete the following substeps.
     1. Select the connection group to which you want to add a new connection and click  .
     2. In the **Add Connection** dialog box, from the **Type** drop-down box, select **Web Service**.
     3. From the **Authentication** drop-down box, select **HMAC** and click **OK**
   - To modify an HMAC web service connection, select the connection you want to modify and either right-click it and select Edit Connection or click  .
3. In the **Add Web Service Connection** or **Edit Web Service Connection** dialog box, in the Name field, type the name of the HMAC web service connection.

4. In the **Description** field, type descriptive text about the web service connection.
5. In the **Client URL** field, provide the beginning portion of the URL that runs the web service. If you have multiple web services that start with the same portion of the URL, you can provide this common portion in the URL field and share the connection among multiple web services.
6. In the **HMAC Authentication** pane, specify the HMAC authentication options by completing the following substeps
    1. In the **Endpoint** field, type the URL extension that is appended to the client URL; this combination specifies the web service endpoint.
    2. From the **HTTP Method** drop-down box, select the HTTP method used (*GET*, *POST*, *PUT*, or *DELETE*).
    3. In the **Content Type** field, type the content type to send to the URL.

    - For JSON authentication service, the content type is *application/JSON*.
    - For XML authentication service, the content type is *application/XML*.

    4. In the **Secret Key** field, type the secret token shared between the client and web service provider used to encrypt the HMAC authorization signature.
    5. In the **Delimiter** field, optionally specify the delimiter used to separate the values in the concatenated string that comprises the HMAC signature.
    6. In the **Header Field** field, type the header field to add to the HMAC signature.
    7. In the **Prefix** field, optionally type text that prefixes the HMAC signature.
    8. In the **Encoding** field, type the text encoding of the HMAC signature.
    9. In the **Algorithm** field, type the algorithm with which to encrypt the HMAC signature.
7. In the pane to the right of the **HMAC Authorization** pane, specify the parameters that comprise the signature by completing one of the following actions.
    - To add a new HMAC parameter, complete the following substeps.
        1. Click  .
        2. In the **New HMAC Parameter** dialog box, in the **Name** field, type the name of the HMAC parameter.
        3. From the **Type** drop-down box, select the type of content to add (*Static String*, *Current Date*, *Content Length*, or *Endpoint*).
            - For **Static String**, type the value of the string you want to include in the **Value** field (this field is dimmed and unavailable for all other types of content).
            - For **Current Date**, Perceptive DataTransfer includes the current date in the HMAC signature.
            - For **Content Length**, Perceptive DataTransfer calculates the content length of the web service payload at runtime and includes it in the HMAC signature.
            - For **Endpoint**, Perceptive DataTransfer includes the web service's endpoint in the HMAC signature.
        4. In the **Position** field, type the numeric position of the parameter to specify the location of the parameter in the HMAC signature.
        5. Click **OK**.

- To modify an HMAC parameter, complete the following substeps.
  1. Select the parameter you want to modify and click  .
  2. In the Edit HMAC Parameter dialog box, from the Type drop-down box, select the type of content to add (Static String, Current Date, Content Length, or Endpoint).
     - For **Static String**, type the value of the string you want to include in the **Value** field (this field is dimmed and unavailable for all other types of content).
     - For **Current Date**, the system includes the current date in the HMAC signature.
     - For **Content Length**, Perceptive DataTransfer calculates the content length of the web service payload at runtime and includes it in the HMAC signature.
     - For **Endpoint**, the system includes the web service's endpoint in the HMAC signature.
  3. Click **OK**.
- To delete an HMAC parameter, complete the following actions.
  1. Select the parameter you want to delete and click  .
  2. In the dialog box, to delete the parameter, click **OK**.

8. If your authentication request requires you to include certain header properties in the request, you can optionally configure request headers information. For example, if you are making an authentication request to the Integration Server, which requires navigating to the appropriate URL and including the Integration Server user name and password in the request header, you can add a request header to the authentication request. Complete one of the following actions.
   - To add a new request header, in the **Request Headers** pane, complete the following substeps.
     1. Click  .
     2. In the **Add Request Header** dialog box, in the **Name** field, type the name of the request header. For example, if you are specifying an Integration Server user name, you could type `X- IntegrationServer-Username`.
     3. In the **Value** field, type the value of the request header. For example, for an X-IntegrationServer-Username request header with *Administrator* as the user name, you would type `Administrator`.
     4. Click **OK**.
   - To modify a request header, complete the following actions.
     1. Select the request header you want to modify in the **Request Headers** pane and click  .
     2. In the **Edit Request Header** dialog box, from the **Value** field, type the value of the request header.
     3. Click **OK**.

- To delete a request header, complete the following actions.
  1. Select the request header you want to delete and click ⛔.
     A dialog box appears, prompting you to confirm that you want to delete the request header.
  2. To delete the request header, click **OK**.

9. Click **OK**.

For example, you can create an HMAC web service connection with the following parameters.

- Client URL: https://api.example.com
- Endpoint: /examplews/application/v1/person/post
- HTTP Method: POST
- Content Type: application/json
- Delimiter: \n
- Header Field: Authorization
- • Prefix: AW2WS C4CA4238A0B923820DCC509A6F75849B:
- Encoding: UTF-8
- Algorithm: HmacSHA1
- HMAC parameters
  - Parameter 1
    - Position: 1
    - Name: Endpoint
    - Type: Endpoint
  - Parameter 2
    - Position: 2
    - Name: Date
    - Type: Current Date
  - Parameter 3
    - Position: 3
    - Name: App ID
    - Type: Static String
    - Value: 7529D7A
  - Parameter 4
    - Position: 4
    - Name: Content-Length
    - Type: Content Length

- Request header
  - Header x-examplews-appid
  - Value7529D7A

Perceptive DataTransfer calculates the HMAC signature by appending the endpoint value (/examplews/application/v1/person/post), the current system date, the static string 7529D7A, and the content length of the payload included in the request. It calculates the signature in this order because the positions of the HMAC parameters are specified as 1 for Endpoint, 2 for Date, and so on.

Perceptive DataTransfer places the delimiter \n between the HMAC parameters to separate them.

It encodes the concatenated string with UTF-8 encoding and encrypts the string with the secret key and the HmacSHA1 encryption algorithm.

Perceptive DataTransfer also appends the prefix, AW2WS C4CA4238A0B923820DCC509A6F75849B, to the beginning of the encrypted string, and the resulting string is placed in the Authorization header.

The web service request includes a request header of x-examplews-appid with the value of 7529D7A. When the web service executes, the web service provider calculates the expected HMAC signature using the same criteria. If the expected HMAC signature matches the HMAC signature passed through the Authorization header, the web service provider completes the request. If they do not match, the web service provider returns an authorization error.

Adding and modify SOAP web service connections

Create a SOAP web service connection when you are using SOAP web services to retrieve or update data. For more information, refer to Configuring SOAP web service data providers.

Note: SOAP web services are supported in Perceptive DataTransfer 6.4.5 and higher.

To add or modify a SOAP web service connection, complete the following steps.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, complete one of the following actions.
   - To add a new connection, complete the following substeps.
     1. Select the connection group to which you want to add a new connection and click  .
     2. In the **Add Connection** dialog box, from the **Type** drop-down box, select **SOAP Web Service**.
   - To modify a web service connection, select the connection you want to modify and click  .
3. In the **Add Web Service Connection** or **Edit Web Service Connection** dialog box, in the **Name** field, type the name of the SOAP web service connection.
4. In the **Description** field, type descriptive text about the SOAP web service connection.
5. In the **URL Endpoint** field, provide the URL endpoint.
6. Click **OK**.

Deleting connections

To delete a connection, complete the following steps.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, select the connection you want to delete and click  .

Testing connections

To test a connection, complete the following steps.

1. Select **Configuration > Connections**.
2. In the **Connection Group Administration** dialog box, select the connection you want to test and click  .
   - If the connection works properly, a dialog box appears and states that the test was successful.
   - If the connection does not work properly, a dialog box appears and indicates why the test failed.

Configuring Procedures

Refer to the following topics for more information:

- Understanding Perceptive DataTransfer Logic
- Understanding Variables
- Configuring Variables
- Configuring Libraries
- Configuring Procedures
- Importing and Exporting Libraries and Procedures

Understanding Perceptive DataTransfer Logic

Perceptive DataTransfer logic is divided into the following sections, which collect procedures related to the process.

- *Match Process* comprises all procedures and flow control related to matching input records to existing records in your database.
- *Review Process* comprises all procedures and flow control related to selecting data from your database for potential matches.
- *Upload Process* comprises all procedures and flow control related to adding or updating information in your database.
- *Match Result Display* logic defines the query that retrieves the information displayed in the Results panel of the Job window when a match occurs. This query retrieves data based on a specified ID value; data columns returned by the query correspond to the columns of the Results pane.

  You use this logic in a procedure and associate it with a header; when a job associated with the header is run, the query returns the information in the Results pane.

For an overview of the steps required to configure jobs that use this query to populate the Results pane in the Job window, refer to Specifying data displayed in Results pane.

- *Database Population Selection* logic contains the query for dynamically extracting input data for a job. You use this logic in a procedure and associate the procedures with a database header; when a job associated with this header is run, the query extracts data from the database source.

Example:

```
begin-procedure test-query
!
begin-select
SSN    "$SSN",
FIRST  "$FIRST",
LAST   "$LAST",
MI     "$MI",
DOB    "+DOB",
SEX    "$SEX",
FROM table_name  end-select
!
end-procedure
```

For an overview of steps for dynamically extracting input data for a job from a database, refer to Obtaining input data from the database.

For information about configuring logic, refer to Appendix A: Programming Concepts. Procedures and libraries are used to organize logic within each option.

Understanding Procedures

Procedures contain the logic that defines how Perceptive DataTransfer determines if there are potential matches between the input data and existing records in the database and how information in the database is added or updated.

Procedures can also be shared and reused through workspaces. To add a procedure shared by another workspace, the workspace version in which you are working must be in edit mode, and the workspace version that contains the procedure you are importing must be in staged or active mode. For more information about workspaces, refer to Configuring Workspaces.

For more information about configuring logic, such as procedure syntax, refer to Appendix A: Programming Concepts.

Understanding Libraries

Libraries are used to group similar types of procedures within a category. For example, for Match Process procedures, you can have the libraries *Match Control* and *Match Display*, and you can group related procedures within these libraries.

Understanding Variables

Refer to the following topics for more information.

- Understanding Data Types
- Understanding Variable Categories
- Understanding Naming Conventions

Understanding Data Types

Perceptive DataTransfer has with three predefined data types for variables and constants.

- Character: Stores string values.
- Date: Stores date values.
- Number: Stores numeric values.

Understanding Variable Categories

Perceptive DataTransfer uses two types of variables:

- Input: Represents each column of an input file.
- Program: User-defined or custom variables.

You should use lowercase text to name program variables to more easily distinguish them from input variables.

Understanding Naming Conventions

Perceptive DataTransfer examines the first character in a variable name to determine the data type of the variable. The data type designators are:

- $: Designates a character variable
- &: Designates a date variable
- #: Designates a numeric variable

Configuring Variables

For definitions of global variables, refer to Understanding global variables.

You can also add variable mappings, which map a program variable to label and to a default value. Refer to Configuring Variable Maps.

Refer to the following topics for more information.

- Adding Variables.
- Adding Missing Variables
- Modifying Variables
- Deleting Variables
- Importing Variables
- Exporting Variables

- Searching Variables
- Finding Unused Variables
- Watching Variables
- Clearing Variables From the Watch List
- Moving Variables to Groups
- Configuring Groups

Adding Variables

Note All input variables, including header variables, must have unique names. To add a variable, complete the following steps.

1. Perform one of the following actions.
   - In the **Job Dashboard**, select the job for which you want to add a variable and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. In the **Program Variables** pane, click the **All tab**.
3. Click  .
4. In the **Add Variable** dialog box, in the Variable name field, type the name of the variable.
5. Click **Apply**.
6. Continue to add variables, as appropriate; when you are finished, click **OK**.

   The variable is added and is displayed in the **Program Variables** pane.

Adding Missing Variables

You can search for missing program variables and add any variables that are referenced in the code but do not exist to Perceptive DataTransfer.

To add missing variables for all procedures, refer to Adding Missing Variables for All Procedures.

To add missing variables for a selected procedure, refer to Adding Missing Variables for a Single Procedure.

Adding Missing Variables for All Procedures

To search for and add missing variables for all Perceptive DataTransfer procedures, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to add a variable and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. In the **Procedures** pane, click  and select **Add Missing Variables**.

   Perceptive DataTransfer searches all procedures for program variables referenced within the procedure code. Any referenced program variables that do not exist are created.

Adding Missing Variables for a Single Procedure

To add missing variables for a single Perceptive DataTransfer procedure, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to add a variable and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Select the procedure for which you want to search for missing variables.
3. In the **Procedure Editor** pane, click  .

Perceptive DataTransfer searches the selected procedure for program variables referenced within the procedure code. Any referenced program variables that do not exist are created.

Modifying Variables

To modify a variable, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to edit a variable and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. In the **Program Variables** pane, click the **All tab**.
3. Select the variable you want to modify and either right-click it and select **Modify** or click  .
4. In the **Edit Variable** dialog box, in the Variable name field, type the name of the variable.
5. Click **Apply**.
6. Click **OK**.

   The name of the variable is modified, and its information appears in the Program Variables pane.

Deleting Variables

To delete a variable, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to delete a variable and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. In the **Program Variables** pane, click the **All tab**.
3. Select the variable you want to delete and either right-click it and select **Delete** or click  .

   The variable is deleted and is removed from the Program Variables pane.

Importing Variables

You can import variables from an XML file by doing the following.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to import a variable and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Click  and select **Import**.
3. In the **Import Variable** dialog box, click the **Browse** button and navigate to the file that you want to import or type the location of the file in the **File** field.
4. Click **OK**.

Exporting Variables

Note You cannot export variables in tablet view.

You can export variables to an XML file by doing the following.

1. Perform one of the following actions.

- In the Job Dashboard, select the job for which you want to export a variable and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
- Select **Configuration > Procedures**.

2. Click  and select **Export**.
3. Save the file to the appropriate location.

Searching Variables

You can search for whole variables or partial text of variables.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to search a variable and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. In the **Program Variables** pane, perform one of the following actions.
   - To search for watched variables, click the **Watch** tab.
   - To search through all variables, click the **All** tab.
3. In the text field in the **Program Variables** pane, type the text for which you want to search. You can use whole words or partial text; if you use partial text, all items starting with the beginning character are returned in the search. Use % as the wildcard character for partial text searches.

   For example, if you search for #c%, all variables that start with #c (including the whole variable #c) are returned in the search.
4. Press **Enter** on your keyboard.

   Matches that are found are displayed in the Program Variables pane. If no matching text is found in the search, the pane does not display any entries.

Finding Unused Variables

You can search all procedures to find references to variables that are not being used. Program variables that appear in procedures in any part of the code or in code that is commented out are considered used variables.

1. Perform one of the following steps.
   - In the Job Dashboard, select the job for which you want to search a variable and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. In the **Program Variables** pane, click  and select **Find Unused**.
3. In the **Unused Program Variables** pane, If you have permission to edit program variables, you can delete unused variables. Perform one of the following actions.
   - To delete a single variable, select it and click  .
   - To delete multiple variables, select the variables by clicking a variable and pressing and holding the **Ctrl** key as you click each variable with your mouse. Next, click  .

Watching Variables

When you run a job in interactive mode and view it in the window that appears when you edit a job, the Program Variables pane displays all program variables. If you have many program variables, to more easily view results for specific variables, you can place such variables in the Watch tab.

To watch a variable, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to watch a variable and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. To watch all program variables referenced within a procedure, click  in the **Procedure Editor**. All referenced variables are added to the **Watch** tab.
3. To watch a single variable or all variables, in the **Program Variables** pane, click the **All** tab.
   - To watch a single variable, select the variable you want to watch and either right-click it and select **Watch** or click  . The variable is moved to the **Watch** tab.
   - To watch multiple variables, select the variables by clicking a variable and pressing and holding the **Ctrl** key as you click each variable with your mouse. Next, click  . The variables are moved to the **Watch** tab.

Clearing Variables From the Watch List

To clear variables from the watch list, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to clear a watch variable and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. In the **Program Variables** pane, click the **Watch** tab.
3. Perform one of the following actions.
   - To clear a single variable, select it and click  .
   - To clear multiple variables, select the variables by clicking a variable and pressing and holding the **Ctrl** key as you click each variable with your mouse. Then click  .
   - To clear all watch variables, either do not select a variable, or select a single variable. Click  .

Moving Variables to Groups

To move a variable to a group, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to move a variable to a group and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. In the **Program Variables** pane, click the **All** tab.
3. Perform one of the following actions.
   - To move a single variable, select the variable that you want to move and either right-click it and select the **Move to Group** or click  .
   - To move multiple variables, select the variables by clicking a variable and pressing and holding the **Ctrl** key as you click each variable with your mouse. Next, click  . The Move Variable to Group dialog box appears.
4. In the **Move Variable to Group** dialog box, from the **Group** drop-down box, select the group to which you want to move the variable.
5. Click **OK**.

   The variable is moved to the specified group.

Configuring Groups

You can use groups to manage variables for similar types. For example, you can create a *Form Variables* group in which to place form variables.

Refer to the following topics for more information.

- Adding Groups
- Renaming Groups
- Deleting Groups

Adding Groups

To add a group, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to add a group and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. In the **Program Variables** pane, click the **Groups** tab.
3. Click  .
4. In the **Add Group** dialog box, in the **Name** field, type the name of the field.
5. Click **OK**.

   The group is added and appears in the Groups tab.

Renaming Groups

To rename a group, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to rename a group and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. In the **Program Variables** pane, click the **Groups** tab.
3. Select the group you want to rename and either right-click it and select **Rename** or click  .
4. In the **Rename Group** dialog box, in the Name field, type the name of the group and click **OK**.

   The name of the group is updated, and its information appears in the Program Variables pane.

Deleting Groups

To delete a group or all groups, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to delete a group and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. In the **Program Variables** pane, click the **Groups** tab.
3. Perform one of the following steps.
   - Select the group you want to delete and either right-click it and select **Delete** or click .
   - To delete all groups, click .
4. In the **Confirm Delete** dialog box, click **Yes** to delete the group.

Configuring Libraries

Refer to the following topics for more information.

- Adding Libraries
- Renaming Libraries
- Specifying Control Libraries
- Deleting Libraries
- Importing and Exporting Libraries and Procedures

Note: In the **Procedures** pane, you can click to expand all buttons in the pane; to collapse them, click .

Adding Libraries

To add a library, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to add a library and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Select the type of procedure to which you want to add a library and either right-click it and select **Add Library** or click .
3. In the **Add Library** dialog box, in the **Library Name** field, type the name of the library.
4. In the **Description** field, type descriptive text about the library.
5. Click **OK**.

   The library is added and appears in the Procedures pane.

Renaming Libraries

To rename a library, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to rename a library and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Select the library you want to rename and either right-click it and select **Rename Library** or click  .
3. In the **Rename Library** dialog box, in the **Library Name** field, type the name of the library.
4. Click **OK**.

   The library is renamed, and its information is refreshed in the Procedures pane.

Specifying Control Libraries

You set a library as the control library when you want the library to control the flow of the process. A control library is the library from which the first code is run for each of the Match, Review, and Upload processes.

To specify control libraries, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to specify a control library and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Select the library you want to set as the control library and either right-click it and select **Control Library** or click  .

   The library is configured as the control library, and the  icon appears next to the library.

Deleting Libraries

Note You cannot delete a library that contains procedures. You must first delete all procedures from the library before deleting the library.

To delete a library, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to delete a library and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Select the library you want to delete and either right-click it and select **Delete Library** or click  . The library is deleted and is removed from the **Procedures** pane.

Configuring Procedures

You can share procedures with other workspaces, and you can import procedures from and export procedures to an XML file.

To open the Procedure Editor, perform one of the following actions.

- Select **Configuration > Procedures** from the main application toolbar.
- In the Job Dashboard, select the job for which you want to configure procedures and then either right-click it and select **Edit** or click the **Edit** button; then, in the window that appears, click the **Procedures** tab.

Refer to the following topics for more information.

- Understanding Procedure Keyboard Shortcuts
- Adding Procedures
- Understanding Procedure Icons
- Configuring Procedure Code
- Copying Procedures
- Renaming Procedures
- Adding and Removing Comments to Lines
- Finding Unused Procedures
- Finding Referenced Procedures
- Activating Procedures
- Deactivating Procedures
- Removing Sharing on Procedures
- Rolling Back Procedures
- Searching Procedures
- Importing and Exporting Libraries and Procedures

Understanding Procedure Keyboard Shortcuts

Refer to the following table for descriptions of shortcut keys for the Procedure Editor.

Shortcut Key	Description
SHIFT+F8	Continue
SHIFT+F6	Next line
CTRL+ALT+F	Find/Replace
CTRL+ALT+W	Watch all variables
CTRL+ALT+M	Add missing variables

Shortcut Key	Description
CTRL+ALT+B	Clear breakpoints
CTRL+1	Toggle comment
CTRL+ALT+S	Save

Adding Procedures

To add a procedure, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to add a procedure and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Select the library to which you want to add a procedure and either right-click it and select **Add Procedure,** or click  .
3. In the **Add Procedures** dialog box, perform one of the following actions.
   - To add a new procedure, complete the following steps.
     3. Click the **Add Procedure** tab.
     4. In the **Procedure Name** field, type the name of the procedure.
     5. In the **Description** field, type descriptive text about the procedure.
   - To add a procedure that is shared by another workgroup, complete the following steps.
     1. Click the **Add Shared Procedure** tab.
     2. From the **Workspace** drop-down box, select the workspace that contains the shared procedure.
     3. In the **Procedure Name** drop-down, select the procedure.

        Note Only procedures that belong to workspaces in staged or active mode are available for sharing.
4. Click **OK**.

The procedure is added and displays in the Procedures pane.

Understanding Procedure Icons

Refer to the following table for a description of the icons that appear next to procedures in the Procedures pane.

Icon	Description
	Procedure is deactivated, has been modified, and is not shared.
	Procedure is shared, has been modified since it was added from the workspace sharing it, and is deactivated.
	Procedure is not shared and has been modified.
	Procedure is not shared, contains errors, and has been modified.
	Procedure is shared and has been modified since it was added from the workspace sharing it.
	Procedure is not shared, contains errors, and has not been modified.
	Procedure has not been modified If you are working in a workspace version in edit mode, the procedure is not shared. If you are working in a workspace version in a mode other than edit mode, this icon displays for all procedures. The sharing status is unknown.
	Procedure is not shared, contains errors, and is deactivated.
	Procedure is not shared and is deactivated.
	Procedure is not shared, has been modified, and is deactivated.
	Procedure is shared.
	Procedure is shared and has since been updated in the workspace sharing it.

Configuring Procedure Code

You configure procedure code in the Procedure Editor, which you can open by doing one of the following.

- In the Job Dashboard, select the job for which you want to add a procedure and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
- Select **Configuration > Procedures**.

For information about configuring logic for procedures, refer to Appendix A: Programming Concepts. Perceptive DataTransfer highlights code syntax as follows.

- Comments, beginning with ! or from ! to the end of the line, appear in green.

```
25      !
26      ! search for a match
27      !
```

- Keywords, which are reserved words such as begin-procedure, begin-select, let, do, and so on, appear blue.

```
1  begin-procedure
2  !
3     begin-select
4        last AS L
5        first AS
6        ssn as SS
7        substr(to
8        substr(to
9        substr(to
10       FROM nu_n
11     WHERE nu_n
12       AND nu_n
13       AND nu_n
14    end-select
15 !
16 end-procedure
```

- Text strings appear in maroon.

```
11      SELECT #nu_id, 'PERS'
```

- Global variables appear in pink.

```
3      ! setup globals
4      let $dbname = _GL.DBNAME_
5      let $username = _GL.USERID_
6      let $f_ban_w = _GL.STATUS_
```

- SKIP, STOP and the on-error=display-error (which appears in the begin-sql line) appear in red.

```
34              if $t_ssn != $f_s
35                  skip SUSPEND,
```

- Errors are highlighted in red. Hover your mouse over the number to display a tooltip with information about the issue.

```
22      if isnull(#s
23          !do setu
E 24    end-if
25      !
26      ! search for
E 27    !
28
29
```

Copying Procedures

To create a duplicate of a procedure, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to copy a procedure and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Select the procedure you want to copy and either right-click it and select **Copy Procedure,** or click  .

   The procedure is copied, and the new procedure is added under the original procedure in the Procedures pane.

Renaming Procedures

To rename a procedure, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to rename a procedure and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Select the procedure you want to rename and either right-click it and select **Rename** or click  . The Rename Procedure dialog box appears.
3. In the **Procedure Name** field, type the name of the procedure.
4. Click **OK**.

   The procedure is renamed and is updated in the Procedures pane. In addition, the first line of code (begin-procedure <*procedure name*>) is updated to reflect the new procedure name.

Adding and Removing Comments to Lines

You can quickly add or remove the comment symbol (!) from the start of selected lines by completing the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to add or remove a comment and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. In the **Procedures** pane, select procedure for which you want to add or remove comments.

3. Select the line or lines to which you want to add or remove comments.
4. In the **Procedure Editor** pane, click . One of the following occurs.
   - If all selected lines start with !, excluding whitespace, the lines are uncommented.
   - If one or more selected lines do not start with !, excluding whitespace, one ! is added to the start of each selected line.

Finding Unused Procedures

You can search all procedures to find references to procedures that are not being used. Control, Match Result, and Data Population procedures are not included in the unused procedures search.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to search a procedure and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. In the **Procedures** pane, click and select **Find Unused Procedures**.
3. In the **Unused Procedures** pane, if you have permission to edit procedures, you can delete unused procedures. Perform one of the following actions.
   - To delete a single procedure, select it and click .
   - To delete multiple procedures, select the procedures by clicking a procedure and pressing and holding the Ctrl key as you click each procedure with your mouse. Next, click .
4. In the dialog box, click **Yes** to delete the procedure.

   If you try to delete procedures for which you do not have edit permissions, those procedures are not deleted.

Finding Referenced Procedures

You can search for procedures that are referenced by other procedures by completing the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to search for a referenced procedure and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. In the **Procedures** pane, select the procedure for which you want to search other procedures for references.
3. Click and select **Referenced By**.

   Perceptive DataTransfer searches all procedures for references to the selected procedure. When the search completes, a dialog box appears and displays the procedures that contain references to the procedure you selected.

Deactivating Procedures

When you deactivate a procedure, the procedure is not used when logic is run; it is skipped. You can reactivate procedures that have been deactivated.

To deactivate a procedure, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to deactivate a procedure and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Right-click the procedure you want to deactivate and select **Deactivate**.

   The procedure is deactivated.

Activating Procedures

To activate a procedure that is deactivated, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to activate a procedure and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Right-click the procedure you want to activate and select **Activate**.

   The procedure is activated.

Removing Sharing on Procedures

To remove sharing on a procedure that is shared with another workspace, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to remove sharing on a procedure and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Select the procedure and either right-click it and select Remove Sharing, or click  .

   Sharing is removed on the procedure.

Rolling Back Procedures

If changes have been made and committed to a procedure, you can use a previous version (*roll back*) of it.

1. Perform one of the following actions
   - In the Job Dashboard, select the job for which you want to roll back a procedure and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Select the procedure you want to roll back and either right-click it and select **Rollback Procedure** or click  .

   The **Rollback Manager** dialog box appears and displays the following information about previous versions of the procedure.
   - Name: Name of the procedure.
   - Status: Status of the procedure (*pending*, *current*, or *dated*).
   - Version Date: Date on which the version was created.
   - Created By: Username of the user who created the version.
   - Comments: Comments that were added about the version.
3. Select the version to which you want to revert and click **Rollback**.
4. In the dialog box, to use the selected version, click **Yes**.

   The procedure is updated to use the selected previous version of the procedure.

Obtaining the Latest Version of Procedures

If changes have been made to and committed to a procedure, you can obtain the latest version of it.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to obtain the latest version of a procedure and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Select the procedure for which you want to obtain the latest version and either right-click it and select **Get Latest** or click  .

   The procedure is updated to the latest version.

Searching Procedures

You can search for information in all procedures or in a specific procedure. Searches use case-sensitive text.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job for which you want to search a procedure and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Perform one of the following actions.
   - To search for information in all procedures, click  in the **Procedures** toolbar.
   - To search for text within a specific procedure, select the procedure and click  in the toolbar of the **Procedure Editor**.
3. In the **Find/Replace** dialog box, complete the following substeps.
   1. In the **Find** field, type the text for which you want to search.
   2. In the **Replace** field, type the text with which you want to replace the text in the Find field.
   3. Perceptive DataTransfer searches all procedures for the text that you entered. If a match is found, the matched text is highlighted in yellow in the **Procedure Editor**. If no matches are found, a dialog box appears and indicates that there are no matches.
   4. To find the next occurrence of the text, click **Next**. When no more matches are found, the first match results are displayed again. If no matches were found, a dialog box appears and indicates that no matches were found.
   5. To replace the found text with the text you specified in the **Replace** field, click **Replace**.

Importing and Exporting Libraries and Procedures

Note You cannot import or export libraries or procedures in tablet view.

You can import procedures and libraries from XML documents, and you can export them to XML documents. You can:

- Export all procedures.
- Export a single procedure.
- Export a single library, which includes all its procedures.
- Import one or more libraries, including their procedures.
- Import a single procedure into a library.

Refer to the following topics for more information.

- Importing Libraries
- Exporting Single Libraries
- Importing Single Procedures
- Exporting Single Procedures

- Exporting All Procedures and Libraries

Importing Libraries

You can import files that contain either the entire procedure tree or a single library and its corresponding procedures.

Note You cannot import libraries in tablet view.

Note XML files that contain a single exported library are named beginning with *LIB* (for example, *LIB55.xml*). XML files that contain all libraries and procedures (the entire procedure tree) are named beginning with *PRCS* (for example, *PRCS987.xml*.) You can import XML files of these types. If you try to import a file type other than a library, a popup message appears that indicates import did not complete because the file type was not appropriate.

1. Perform one of the following actions.
   - In the Job Dashboard, select a job and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Click  and select **Import Libraries**.
3. In the **Import Libraries** dialog box, click the **Browse** button and navigate to the XML file that contains the libraries you want to import, or type the location of the file in the File field.
4. Check the **Update Existing Entities** check box to update an existing library, with the same name as the library you are importing, with the library you are importing.

   If you do not check this check box, and there is an existing library with the same name as the library you are importing, *DUPLICATE-* is appended to the start of the name of the library you are importing; the existing library is not updated. If there is a library with the name DUPLICATE-<*library_name*>, *DUPLICATE(2)-* is appended to the start of the name. The number continues to be incremented so that all libraries have unique names.
5. Click **OK**.

   The Import Report appears and displays information about the import.

Exporting Single Libraries

Note You cannot export single libraries in tablet view. To export a single library, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select a job and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Select the appropriate library.
3. Click  and select **Export**.
4. Save the file in the appropriate location.

   XML files that contain libraries are named starting with *LIB* (for example, *LIB55.xml*). You can import files of this type when you are importing libraries (when you import a single library and all its procedures).

Importing Single Procedures

You cannot import single procedures in tablet view.

Note XML files that contain a single exported procedure are named beginning with *PROC* (for example, *PROC123.xml*). You can import XML files of this type. If you try to import a file type other than a single procedure, a popup message appears that indicates import did not complete because the file type was not appropriate.

To import a single procedure, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select a job and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Select the library into which you want to import the procedure.
3. Click and select **Import**.
4. In the **Import Procedure** dialog box, click the **Browse** button and navigate to the file that contains the procedure you want to import, or type the location of the file in the **File** field.
5. Check the **Update Existing Entities** check box to update an existing procedure, with the same name as the procedure you are importing, with the procedure you are importing.

   If you do not check this check box, and there is an existing procedure with the same name as the procedure you are importing, *DUPLICATE-* is appended to the start of the name of the procedure you are importing; the existing procedure is not updated. If a procedure exists with the name DUPLICATE-<*procedure_name*>, *DUPLICATE(2)-* is appended to the start of the name. The number continues to be incremented so that all procedures have unique names.
6. Click **OK**.

   The Import Report appears and displays information about the import.

Exporting Single Procedures

Note You cannot export single procedures in tablet view. To export a single procedure, complete the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select the job a job and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Select the procedure you want to export.
3. Click and select **Export**.
4. Save the file in the appropriate location.

   XML files that contain a single exported procedure are named beginning with *PROC* (for example, *PROC123.xml*). You can import files of this type when you import a single procedure.

Exporting All Procedures and Libraries

Note You cannot export all procedures and libraries in tablet view.

You can export all procedures and libraries (the entire procedure tree) as an XML file by completing the following steps.

1. Perform one of the following actions.
   - In the Job Dashboard, select a job and either right-click it and select **Edit** or click the **Edit** button. In the window that appears, click the **Procedures** tab.
   - Select **Configuration > Procedures**.
2. Perform one of the following actions.
   - Without clicking any selection in the Procedures pane, click  and select **Export**.
   - Select a procedure type node (such as Match Process), library, or procedure; then, click  and select **Export All**.
3. Save the file in the appropriate location.

   XML files that contain all exported procedures are named beginning with *PRCS* (for example, *PRCS987.xml*). You can import files of this type when you import libraries (when you import the entire procedure tree).

Configuring Variable Maps

You can use variable maps to map program variables to a label and description. You can also use variable maps to provide a default value for a variable.

You can also use the InitializeVars function with variable maps. Refer to Appendix A: Programming Concepts.

Refer to the following topics for more information.

- Adding and Modifying Variables Maps
- Deleting Variable Maps
- Copying Variable Maps
- Removing Sharing on Variable Maps
- Rolling Back Variable Maps
- Obtaining the Latest Versions of Variable Maps
- Exporting Variable Maps
- Importing Variable Maps

Adding and Modifying Variables Maps

To add a variable mapping, complete the following steps.

1. Select **Configuration > Variable Maps**.
2. In the **Add Variable Mapping Administration** dialog box, perform one of the following actions.
   - To add a new variable map, click  to display the **Add Variable Mapping** dialog box.
     6. To add a new variable map, click the **Add Variable Map** tab. Then, in the **Name** field, type the name of the variable map.
     7. To add a variable map that another workspace is sharing, click the **Add Shared Variable Map** tab. Then, from the Workspace drop-down box, select the workspace that is sharing the variable map, and select the variable map from the Variable Map drop-down box.
     8. Click **OK**.
   - To modify a variable map, select the variable map and either right-click it and select **Edit** or click  .
     1. In the **Edit Variable Mapping** dialog box, in the **Name** field, you can modify the name of the variable map by typing a name.
     2. In the **Description** field, you can modify the description by typing descriptive text.
     3. In the **Program Variables** pane, you can perform actions such as add, modify, watch, import, and export variables. Refer to Configuring Variables for more information.
3. To add a mapping between a variable and a label or default value, complete the following substeps.
     1. In the rightmost pane, click  .
     2. In the **New Variable Mapping** dialog box, in the **Name** field, type the label you want to assign to the variable.
     3. In the **Description** field, optionally type descriptive text about the variable mapping.
     4. In the **Variable** field, optionally type the program variable you want to map.
     5. In the **Default** field, optionally type the default value to which you want to map the program variable.
     6. Click **OK**.
4. To modify a variable mapping, perform one of the following actions.
   - If you are using desktop view, in the rightmost pane, double-click the cell you want to modify and type the appropriate value. You can also modify the variable by clicking the entry you want to modify in the rightmost pane and dragging and dropping a variable from the Program Variables pane to the rightmost pane.
   - If you are using tablet view, complete the following steps.
     1. In the rightmost pane, click  .
     2. In the **New Variable Mapping** dialog box, in the **Name** field, type the label you want to assign to the variable.
     3. In the **Description** field, optionally type descriptive text about the variable mapping.
     4. In the **Variable** field, optionally type the program variable you want to map.

5. In the **Default** field, optionally type the default value to which you want to map the program variable.
6. Click **OK**.

5. To delete a variable mapping, select the mapping you want to delete in the rightmost pane and click  .

Deleting Variable Maps

To delete a variable map, complete the following steps.

1. Select **Configuration > Variable Maps**.
2. In the **Variable Map Administration** dialog box, select the variable map you want to delete and either right-click it and select Delete or click  .

   The variable map is deleted and is removed from the Variable Map Administration dialog box.

Copying Variable Maps

To create a duplicate of a variable map, complete the following steps.

1. Select **Configuration > Variable Maps**.
2. In the **Variable Map Administration** dialog box, select the variable map you want to copy and either right-click it and select **Copy** or click  .

   A copy of the variable map is created and appears in the Variable Map Administration dialog box.

Removing Sharing on Variable Maps

To remove sharing on a variable map that is shared with another workspace, complete the following steps.

1. Select **Configuration > Variable Maps**.
2. In the **Variable Map Administration** dialog box, select the variable for which you want to remove sharing and either right-click it and select **Remove Sharing**, or click  .

   Sharing is removed on the variable map.

Rolling Back Variable Maps

If changes have been made and committed to a variable map, you can use a previous version (*roll back*) of it.

1. Select **Configuration > Variable Maps**.
2. In the **Variable Map Administration** dialog box, click the variable map you want to roll back and either right-click it and select **Rollback,** or click  . The **Rollback Manager** dialog box appears and displays the following information.
   - Name: Name of the variable map.
   - Status: Status of the variable map (*current* or *dated*).
   - Version Date: Date and time on which the version was created.

- Created By: Username of the user who created the variable map.
- Comments: Comments that were added by the user.

3. Select the previous version to which you want to revert and click **Rollback**.

   The previous version is loaded.

Obtaining the Latest Versions of Variable Maps

If changes have been made to and committed to a variable map, you can obtain the latest version of it.

1. Select **Configuration > Variable Maps**.
2. In the **Variable Map Administration** dialog box, click the variable map for which you want to obtain the latest version and either right-click it and select **Get Latest** or click  .

Exporting Variable Maps

Note You cannot export variable maps in tablet view.

You can export a variable map to an XML file by doing the following.

1. Select **Configuration > Variable Maps**.
2. In the **Variable Map Administration** dialog box, select the variable map you want to export.
3. Click  and select **Export**.
4. Save the file to the appropriate location.

Importing Variable Maps

Note You cannot import variable maps in tablet view.

1. Select **Configuration > Variable Maps**.
2. In the **Variable Map Administration** dialog box, click  and select **Import**.
3. In the **Variable Map** dialog box, click the **Browse** button and navigate to the file that you want to import or type the location of the file in the **File** field.
4. Check the **Update Existing Entities** check box to update an existing variable map, with the same name as the variable map you are importing, with the variable map you are importing.

   If you do not check this check box, and there is an existing variable map with the same name as the variable map you are importing, *DUPLICATE-* is appended to the start of the name of the variable map you are importing; the existing variable map is not updated. If a variable map exists with the name DUPLICATE-*<index_map_name>*, *DUPLICATE(2)-* is appended to the start of the name. The number continues to be incremented so that all variable maps have unique names.
5. Click **OK**.

Configuring EDI formats

You can process electronic data interchange (EDI) files in Perceptive DataTransfer. Perceptive DataTransfer manages EDI formats as XML files. An EDI format defines the rules by which the EDI input file is converted to XML. You must specify an EDI format for each EDI input file you want to process. After you define the EDI format, configure an EDI header and use it for the appropriate job that processes the EDI input file.

Note EDI formats are supported in Perceptive DataTransfer 6.4.4 and higher. Refer to the following topics for more information.

- Understanding EDI files
- Adding and modifying EDI formats
- Deleting EDI formats
- Copying EDI formats
- Verifying EDI formats
- Removing sharing on EDI formats
- Rolling back EDI Formats
- Obtaining the latest versions of EDI formats
- Exporting EDI formats
- Importing EDI formats

Understanding EDI files

An EDI file begins with a segment ID (a prefix of two or three letters, such as *ISA*). Each line of a file is a string that contains fields, separated by a delimiter. Segments can be nested. EDI segments translate to XML tags or nodes; the fields within a segment translate into XML leaf nodes. With an EDI format, you define the nesting rules and leaf nodes so that Perceptive DataTransfer can construct the corresponding XML.

For example:

```
ISA*00!      !00!   !ZZ!TXAPP     !22!003560

!080906!2015!U!00402!130906001!0!T!$\
GS!AF!TXAPP!DBU!20080906!1257!080906001!X!004020ED0020\

ST!189!062180001\

BGN*00*062180001!20080906*1257!CT\

N1!TM!!ZZ!TXAPP\

N1!AT!!73!003560\

REF!48!0800000638!FRESHMAN APPLICATION ID\

DTP!196!CM!200809\

REF!SY!!STUDENT DECLINED TO PROVIDE SSN\

REF!ZZ!A123456789BC!I-551 NUMBER\

DTP!102!D8!19991001\
```

```
IN1!1!02\
IN2!05!TEST\
IN2!02!FIRST\
IN2!03!M\
IN2!09!JR\
```

In the preceding example, prefixes include *ISA*, *GS*, and *ST*. The EDI file includes *!,* \, and * as delimiters.

IN2 is the child segment of *IN1*; the IN1 segment begins an IN1 loop, which can contain a various assortment of its possible segment children. The file translates logically into the following XML format.

```
<ISA>
   ....
   <GS>... </GS>
   <ST>... </ST>
   ...
   <IN1>
      ....
      <IN2>... </IN2>
      <IN2>... </IN2>
      ...
   </IN1>
   ...
</ISA>
```

The segments translate to XML tags and the fields within a segment translate into XML leaf nodes.

```
<IN2>
   <field1>1</field1>
   <field2>102</field2>
</IN2>
```

Adding and modifying EDI formats

To add or modify an EDI format, complete the following steps.

1. Select **Configuration > EDI Formats**.
2. In the **EDI Format Administration** dialog box, complete one of the following actions.
   - To add a new EDI format, click  and complete the following actions.

     In the **Add EDI Format** dialog box, complete one of the following actions.
     - To add a new EDI format, click the **Add EDI Format** tab. In the **Name** field, type the name of the EDI format.

- To add an EDI format that another workspace is sharing, click the **Add Shared EDI Format** tab. From the **Workspace** list, select the workspace that is sharing the EDI format and select the EDI format from the **EDI Format** list.

  Click **OK**.

- To modify an EDI format, select the EDI format and click .

3. In the **Edit EDI Format <*Format_name*>** dialog box, in the **Name** box, type a name to modify the name of the EDI format.
4. In the **Description** box, enter a description for the EDI format.
5. In the **Delimiters** box, type the file delimiters (for example, *!)*. You can provide multiple delimiters, separated by commas.
6. Add or modify child segments by completing the following substeps.

    1. Complete one of the following actions.

    - To add a child segment, in the **Child Segments** pane, click .
    - To modify a child segment, select the segment and click .

    2. In the **Position** box, type the position of the segment. The position is used for display purposes only; ordering is not relevant to the XML format.
    3. In the **Segment** box, type the prefix that identifies the segment. It must match the segment ID as it appears in the **EDI** input file.
    4. In the **Alias** box, type the alternate name used for the segment.
    5. The **Fields** pane displays the fields contained by the segment. To add or modify a field, complete the following substeps.

    1. Complete one of the following actions.

    - To add a field, click .
    - To modify a field, select the field and click .

    2. In the **Name** field, type the name of the field. Field names must be unique for each node.
    3. In the **Order** field, type the order or position of the field as it appears in the segment. Each field must have a unique order.
    4. Click **Apply** to apply your changes. You can continue to add or edit fields in the dialog box.
    5. Click **OK** to close the dialog box.
    6. To delete a field, select the field and click .
    7. You can continue to add child segments to the segment in the **Child Segments** pane and then add and modify fields for the child segment, as appropriate. The dialog box caption for the **Edit *EDI_Format*** displays the path of the segment so that you can identify your location.

7. To delete a child segment, select the segment you want to delete and either right-click it and select **Delete** or click .
8. Create an EDI header that uses the EDI format you defined. Refer to Configuring and modifying EDI headers and header variables Configuring and modifying EDI.

Deleting EDI formats

To delete an EDI format, complete the following steps.

1. Select **Configuration > EDI Formats**.
2. In the **EDI Format Administration** dialog box, select the EDI format you want to delete and click .

Copying EDI formats

To create a duplicate of an EDI format, complete the following steps.

1. Select **Configuration > EDI Formats**.
2. In the **EDI Format Administration** dialog box, select the EDI format you want to copy and click .

Verifying EDI formats

To verify that your EDI input file converts to XML properly, complete the following steps.

1. Select **Configuration > EDI Formats**.
2. In the **EDI Format Administration** dialog box, select the EDI format you want to test and click .
3. In the **EDI to XML File** dialog box, click the **Browse** button to navigate to the EDI input file you want to test.
4. Click **OK**.

   Perceptive DataTransfer converts the EDI file to an XML file and streams it to your browser.

Removing sharing on EDI formats

To remove sharing on an EDI format that is shared with another workspace, complete the following steps.

1. Select **Configuration > EDI Formats**.
2. In the **EDI Format Administration** dialog box, select the EDI format for which you want to remove sharing and click .

Rolling back EDI Formats

If changes have been made and committed to an EDI format, you can roll back to a previous version.

1. Select **Configuration > EDI Formats**.
2. In the **EDI Format Administration** dialog box, select the EDI format you want to roll back and either right-click it and select Rollback or click .

   The **Rollback Manager** dialog box displays the following information.

   - Name: Name of the EDI format.
   - Status: Status of the EDI format (*current* or *dated*).
   - Version Date: Date and time the version was created.

- Created By: User name of the user who created the EDI format.
- Comments: Comments that were added by the user.

3. Select the previous version to which you want to revert and click **Rollback**.

Obtaining the latest versions of EDI formats

If changes have been made and committed to an EDI format, you can obtain the latest version of it.

1. Select **Configuration > EDI Formats**.
2. In the **EDI Format Administration** dialog box, click the EDI format for which you want to obtain the latest version and click  .

Exporting EDI formats

To export an EDI format to an XML file, complete the following steps.

Note You cannot export EDI formats in tablet view.

1. Select **Configuration > EDI Formats**.
2. In the **EDI Format Administration** dialog box, select the EDI format you want to export.
3. Click  and select **Export**.
4. Save the file to the appopriate location.

Importing EDI formats

To import EDI formats, complete the following steps.

Note You cannot import EDI formats in tablet view.

1. Select **Configuration > EDI Formats**.
2. In the **EDI Format Administration** dialog box, click  and select **Import**.
3. In the **Import EDI Format** dialog box, click the **Browse** button and navigate to the file that you want to import or type the location of the file in the **File** field.
4. Check the **Update Existing Entities** check box to update an existing EDI format, with the same name as the EDI format you are importing, with the EDI format you are importing.

   If you do not check this check box, and there is an existing EDI format having the same name as the EDI format you are importing, *DUPLICATE-* is appended to the start of the name of the EDI format you are importing and the existing EDI format is not updated. If an EDI format exists with the name DUPLICATE-<*EDI_format_name*>, *DUPLICATE(2)-* is appended to the start of the name. The number continues to be incremented so that all EDI formats have unique names.
5. Click **OK**.

Configuring headers

Refer to the following topics for more information.

- Understanding headers
- Understanding header icons
- Adding headers
- Configuring and modifying headers and header variables
- Removing sharing on headers
- Rolling back headers
- Obtaining the latest versions
- Deleting headers
- Exporting headers
- Importing headers

Understanding headers

Headers define the import source, which is the input file with the data you will be loading. For example, you may want to import student SAT scores and load them into your database, either linking the test scores to existing records or creating new records of information. If that data is contained in a spreadsheet of information in CSV format, you are using a delimited input file.

With Perceptive DataTransfer, you map a data input source to a destination, usually mapping a column from an input file to a database table column. Header variables comprise headers, which are input variables that identify the field element and field order within the import file.

You can share headers with other workspaces, and you can import headers from and export headers to an XML file.

Understanding header icons

The following table contains information about the header icons that appear in the Header Administration dialog box.

Icon	Description
	Header is shared.
	Header is shared and has been modified since it was added from the workspace sharing it.
	Header is not shared and has been modified.
	Header has not been modified.

Adding headers

To add a new header, complete the following actions.

1. Select **Configuration > Headers**.
2. In the **Header Administration** dialog box, click  .
3. Complete one of the following actions.
   - To add a new header, click the **Add Header** tab.
     6. In the **Header Name** field, type the name of the header.
     7. From the **Type** drop-down box, select the type of header.
     - Delimited
     - Fixed-Width
     - MS Excel
     - Database
     - XML
     - EDI
     - Hobsons Connect CRM
     - Web Service

       Note: EDI headers are supported in Perceptive DataTransfer 6.4.4 and higher.
   - To add a header that is shared by another workspace, click the **Add Shared Header** tab.
     1. From the **Workspace** drop-down box, select the workspace that shares the header you want to use.
     2. From the **Header** drop-down box, select the header to use.
4. Click **OK**.
5. Configure the header and add header variables. Refer to Configuring and modifying headers and header variables.

Configuring and modifying headers and header variables

Refer to the following topics for more information.

- Configuring and modifying fixed-width headers and header variables
- Configuring and modifying delimited headers and header variables
- Configuring and modifying database headers
- Configuring and modifying Microsoft Excel headers and header variables
- Configuring and modifying XML headers and header variables
- Configuring and modifying EDI headers and header variables
- Configuring Hobsons Connect CRM headers and header variables
- Configuring and modifying web service headers

Configuring and modifying fixed-width headers and header variables

To configure a fixed-width header, complete the following steps.

1. Select **Configuration > Headers**.
2. Click the fixed-width header you want to modify and click .
3. In the **Header Editor** dialog box, in the **Header Information** pane, complete the following substeps.
    1. Type the name of the header in the **Header Name** field.
    2. From the **Match Result Display** drop-down box, select the procedure that defines the information that displays in the **Results** pane in the Job window when a match occurs. Refer to Configuring Procedures for more information about procedures.

       Note: For an overview of the steps required to configure jobs that use this query to populate the Results pane in the Job window, refer to Specifying data displayed in Results pane.
    3. In the **File Properties** pane, select the appropriate check box.
        - **Exclude First Row**: Select this option to exclude the first row in the file when processing the file. By default, this check box is not selected.
        - **Trim Whitespace**: Select this option to remove the whitespace of data fields when processing the data. By default, this check box is selected.
4. In the **Header Variables** pane, configure the header variables.
5. To add a new variable, click in the **Header Variables** pane.
    - If you do not click a variable in the table, the new variable appears as the first entry in the table.
    - If you click a variable, the new variable is added below the selected variable.
6. To modify a header variable, complete the following substeps.
        1. In the **Header Variables** pane, select the variable you want to modify and either right-click it and select **Edit** or .
        2. In the **Edit Input Variable** pane, in the **Position** field, type the number of the position of the column. You may need to adjust the column position if a data file changes.

       Use this field to move the column to a new position and to adjust all other columns to the left or to the right, accordingly.
        3. In the **Name** field, type the name of the variable.
        4. From the **Data Type** drop-down box, select the data type of the variable (*Character*, *Number*, or *Date*).

       In the **Name** field, the appropriate symbol used to specify the data type of the variable is appended to the first character of the name.

       The **$** (dollar sign) denotes a character variable, **#** (number sign) denotes a numeric variable, and **&** (ampersand) denotes a date variable.
        5. Select the **Display** check box to display the column in the pane that displays records in the window that appears when you open a job.
        6. Check the **Sort** check box to allow this column to be sorted in the **Records** panes in the **Edit Job** window (**Find** click a job in the **Job Dashboard** and either click **Edit** or right-click it and select **Edit**) or in the window that appears when a job is opened in the **Job**

Dashboard (which appears when you select a record and either click **Open** or right-click it and select **Open**) You can select up to ten columns to be sorted.

Note A column that is set as the record key is also a sortable column and counts toward the ten elements that can be sortable.

Sorting is based on the prefix of the input variable. If the prefix is **&**, sorting is based on timestamp (which in the end is a numeric sort); if the prefix is **$**, sorting is alphanumeric; and if the prefix is **#**, sorting is numeric.

These values are all stored as strings regardless of the prefix the values are converted to when the user opens or edits a job to prepare a job. If, for example, the prefix is # and the value does not convert to a numeric value, the value is managed as *null*; for #ZIP and value SEVEN, seven is *null* when sorted.

7. In the **Tooltip** field, type the text that you want to appear when you hover your mouse over the column in the pane that displays records that appear when you open a job.
8. In the **Start** field, type the start position of the column. Note the following information.

- The end position is adjusted accordingly.
- The width is not changed.
- The size of the columns next to the column (previous or next) are adjusted accordingly.
- You cannot change the start position to be larger than the end position.
- You cannot change the start position of a column that has a position of *1*.
- When you move the start position to the left:
  - If you move the start position to or before the start of any other column, the start position of the other columns are moved to the right accordingly.
  - If you move the start position to between the start and end range of another column, the width and end position of the other column are changed, making the column smaller in size.
- When you move the start position to the right:
  - The column width is unchanged.
  - The end position is adjusted accordingly.
  - The width and end positions of the previous column are changed (making the column larger in size).

9. In the **End** field, type the end position of the column. Note the following information.

- The end position cannot be before the start position.
- When you move the end to the left:
  - The width of the column becomes smaller in size.
  - The start position of the next column is adjusted.
- When you move the end to the right:
  - The width of the column becomes larger in size.
  - The start position of the next column is adjusted.

10. In the **Width** field, type the width of the column. Note the following:

- The end position is adjusted accordingly.
- The start position of the next column is adjusted.

11. Click **OK**.

7. To prevent duplicate records when this header is used to run jobs, you can set a column as the record key. Jobs run more quickly when a column is set as a record key.

   Selecting a record key allows Perceptive DataTransfer to process any records with different values in the selected column at the same time. You do not need to select a column that guarantees unique values, but the column should capture duplicate records. For example, if you select Last Name as the key to a header and process a file that has many records with the last name Smith, which represent different students, you prevent any two records with the same last name from being processed at the same time.

   To mark a column as the record key, select the column and click  . The  icon appears next to the column to denote that it is the record key. You can mark one column only as the record key. You cannot configure composite keys.

   Note A column that is set as the record key is also a sortable column and counts toward the ten elements that can be sortable.

8. To delete a variable, select it and click  .
9. To preview data, in the **Data Preview** pane, click  .
10. In the **Upload Preview File** dialog box, click **Browse** and navigate to the file that you want to preview, and then select it.
11. Click **Preview** to view the data in the **Data Preview** pane.
12. To clear data from the pane, click 

Configuring and modifying delimited headers and header variables

To configure a delimited header, complete the following steps.

1. Select **Configuration > Headers**.
2. In the **Header Administration** dialog box, click the delimited header you want to modify and click  .
3. In the **Header Editor** dialog box, in the **Header Information** pane, complete the following substeps.
    1. Type the name of the header in the **Header Name** field.
    2. From the **Match Result Display** drop-down box, select the procedure that defines the information that displays in the **Results** pane in the **Job** window when a match occurs. Refer to Configuring Procedures for more information about procedures.

    For an overview of the steps required to configure jobs that use this query to populate the Results pane in the **Job** window, refer to Specifying data displayed in Results pane.

4. In the **File Properties** pane, select the appropriate check box:
    - **First Row Column Names**: Select this option to load the first row of the data file into the file header that is used for column names.
    - **Delimited**: Select this option to specify that the input file is opened as a delimited file. Then click the appropriate option.
        - **Tab**: Select this option to specify that the file is opened as a tab-delimited file.
        - **Specify**: Select this option to set a value by which the data is delimited in the field and provide the value in the **Specify** field.
        - **Exclude First Row**: Select this option to exclude the first row in the file when processing the file. By default, this check box is not selected.
        - **Trim Whitespace**: Select this option to remove the whitespace of data fields when processing the data. By default, this check box is selected.
5. To add a new variable, click  in the **Header Variables** pane.
    - If you do not click a variable in the table, the new variable appears as the first entry in the table.
    - If you click a variable, the new variable is added below the selected variable.

6. To modify a variable, complete the following substeps.
    1. In the **Header Variables** pane, select the variable you want to modify and click  .
    2. In the **Edit Input Variable** dialog box, in the **Position** field, type the number of the position of the column. You may need to adjust the column position if a data file changes.
    3. In the **Name** field, type the name of the variable.
    4. From the **Data Type** drop-down box, select the data type of the variable (*Character*, *Number*, or *Date*).

    In the **Name** field, the appropriate symbol used to specify the data type of the variable is appended to the first character of the name.

    The **$** (dollar sign) denotes a character variable, **#** (number sign) denotes a numeric variable, and **&** (ampersand) denotes a date variable.

    5. Select the **Display** check box to display the column in the pane that displays records in the window that appears when you open a job.
    6. Select the **Sort** check box to allow this column to be sorted in the **Records** pane in the **Edit Job** window (**Find** a job in the **Job Dashboard** and either click **Edit** or right-click it and select **Edit**) or in the window that appears when a job is opened in the **Job Dashboard** (which appears when you select a record and either click **Open** or right-click it and select **Open**) You can select up to ten columns to be sorted.

    Note: A column that is set as the record key is also a sortable column and count toward the ten columns that can be sortable.

    Sorting is based on the prefix of the input variable. If the prefix is **&**, sorting is based on timestamp (which in the end is a numeric sort); if the prefix is **$**, sorting is alphanumeric; and if the prefix is **#**, sorting is numeric.

    These values are all stored as strings regardless of the prefix the values are converted to when the user opens or edits a job to prepare a job. If, for example, the prefix is # and the value does not convert to a numeric value, the value is managed as *null*; for #ZIP and value SEVEN, seven is *null* when sorted.

    7. In the **Tooltip** field, type the text that appears when you hover your mouse over the column in the pane that displays records that appear when you open a job.
    8. Click **OK**.
7. To prevent duplicate records when this header is used to run jobs, you can set a column as the record key. Jobs run more quickly when a column is set as a record key.

    Selecting a record key allows Perceptive DataTransfer to process any records with different values in the selected column at the same time. You do not need to select a column that guarantees unique values, but the column should capture duplicate records. For example, if you select *Last Name* as the key to a header and process a file that has many records with the last name *Smith*, which represent different students, you prevent any two records with the same last name from being processed at the same time.

To mark a column as the record key, select the column and click . The icon appears next to the column to denote that it is the record key. You can mark one column only as the record key. You cannot configure composite keys.

Note A column that is set as the record key is also a sortable column and counts toward the ten elements on which you can sort.

8. To delete a variable, select it and click .
9. Click . in the **Data Preview** pane
10. In the Upload Preview File dialog box, click **Browse** and navigate to the file that you want to preview, and then select it.
11. Click **Preview** to view data in the **Data Preview** pane.
12. To clear data from the pane, click .

Configuring and modifying database headers

You can use database headers to dynamically extract input data from the database and use it when a job is run. For an overview of steps you must perform to dynamically obtain input data from the database for a job, refer to Obtaining input data from the database.

To configure a database header, complete the following steps.

1. Select **Configuration > Headers**.
2. In the **Header Administration** dialog box, click the database header you want to modify and click .
3. In the **Header Editor** dialog box, in the **Header Information** pane, complete the following steps.
    1. Type the name of the header in the **Header Name** field.
    2. From the **Match Result Display** drop-down box, select the procedure that defines the information that displays in the **Results** pane in the **Job** window when a match occurs. Refer to Configuring Procedures for more information about procedures.

    For an overview of the steps required to configure jobs that use this query to populate the **Results** pane in the Job window, refer to Specifying data displayed in Results pane.

    3. In the **Population Selection** field, select the Database Population Selection procedure that contains the query that dynamically extracts the input data. The query that captures the header input variables and displays them in the **Header Variables** pane. Refer to Configuring Procedures for more information about procedures.

    For an overview of the steps required to configure jobs into which input data is automatically extracted, refer to Obtaining input data from the database.
4. If you selected a query in the **Population Selection** field, to capture header variables from the query, click **Capture**. The header variables display in the **Header Variables** pane.

    Note Perceptive DataTransfer uses the aliases in the selected Database Population Selection procedure to create and store column headings (input variables). You cannot edit these input variables.
5. To edit a header variable, complete the following substeps.
    1. In the H**eader Variables** pane, select the variable you want to modify and click .

2. In the **Edit Input Variable** dialog box, in the **Position** field, type the number of the position of the column. You may need to adjust the column position if a data file changes.
3. In the **Name** field, type the name of the variable.
4. From the **Data Type** drop-down box, select the data type of the variable (*Character*, *Number*, or *Date*).

In the **Name** field, the appropriate symbol used to specify the data type of the variable is appended to the first character of the name.

The **$** (dollar sign) denotes a character variable, # (number sign) denotes a numeric variable, and **&** (ampersand) denotes a date variable.

5. Select the **Display** check box to display the column in the pane that displays records in the window that appears when you open a job.
6. Select the **Sort** check box to allow this column to be sorted in the **Records** pane in the **Edit Job** window (**Find** a job in the **Job Dashboard** and either click **Edit** or right-click it and select **Edit**) or in the window that appears when a job is opened in the **Job Dashboard** (which appears when you select a record and either click **Open** or right-click it and select **Open**) You can select up to ten columns to be sorted.

Note: A column that is set as the record key is also a sortable column and count toward the ten columns that can be sortable.

Sorting is based on the prefix of the input variable. If the prefix is **&**, sorting is based on timestamp (which in the end is a numeric sort); if the prefix is **$**, sorting is alphanumeric; and if the prefix is #, sorting is numeric.

These values are all stored as strings regardless of the prefix the values are converted to when the user opens or edits a job to prepare a job. If, for example, the prefix is # and the value does not convert to a numeric value, the value is managed as *null*; for #ZIP and value SEVEN, seven is *null* when sorted.

7. In the **Tooltip** field, type the text that appears when you hover your mouse over the column in the pane that displays records that appear when you open a job.
8. Click **OK**.

6. To prevent duplicate records when this header is used to run jobs, you can set a column as the record key. Jobs run more quickly when a column is set as a record key.

   Selecting a record key allows Perceptive DataTransfer to process any records with different values in the selected column at the same time. You do not need to select a column that guarantees unique values, but the column should capture duplicate records. For example, if you select *Last Name* as the key to a header and process a file that has many records with the last name *Smith*, which represent different students, you prevent any two records with the same last name from being processed at the same time.

   To mark a column as the record key, select the column and click  . The  icon appears next to the column to denote that it is the record key. You can mark one column only as the record key. You cannot configure composite keys.

   Note A column that is set as the record key is also a sortable column and count toward the ten columns that can be sortable.

7. To delete a variable, select it and click  .

Configuring and modifying Microsoft Excel headers and header variables

Note Microsoft Excel Headers support files created in Excel versions 97 and later. To configure a Microsoft Excel header, complete the following actions.

1. Select **Configuration > Headers**.
2. In the **Header Administration** dialog box, click the Microsoft Excel header you want to modify and click  .
3. In the **Header Editor** dialog box, in the **Header Information** pane, complete the following substeps.
    1. Type the name of the header in the **Header Name** field.
    2. From the **Match Result Display** drop-down box, select the procedure that defines the information that displays in the Results pane in the **Job** window when a match occurs. Refer to Configuring Procedures for more information about procedures.

    For an overview of the steps required to configure jobs that use this query to populate the **Results** pane in the **Job** window, refer to Specifying data displayed in the Results pane.

    3. In the **File Properties** pane, select the appropriate check box:

- First Row Column Names: Select this option to load the first row of the data file into the file header that is used for column names.
- Exclude First Row: Select this option to exclude the first row in the file when processing the file. By default, this check box is not selected.
- Trim Whitespace: Select this option to remove the whitespace of data fields when processing the data. By default, this check box is selected.

4. To add a new variable, click  in the **Header Variables** pane.
   - If you do not click a variable in the table, the new variable appears as the first entry in the table.
   - If you click a variable, the new variable is added below the selected variable.
5. To edit a header, complete the following substeps.

   1. In the **Header Variables** pane, select the variable you want to modify and click  .
   2. In the **Edit Input Variable** dialog box, in the **Position** field, type the number of the position of the column. You may need to adjust the column position if a data file changes.
   3. In the **Name** field, type the name of the variable.
   4. From the **Data Type** drop-down box, select the data type of the variable (*Character*, *Number*, or *Date*).

   In the **Name** field, the appropriate symbol used to specify the data type of the variable is appended to the first character of the name.

   The **$** (dollar sign) denotes a character variable, # (number sign) denotes a numeric variable, and **&** (ampersand) denotes a date variable.

   5. Select the **Display** check box to display the column in the pane that displays records in the window that appears when you open a job.
   6. Select the **Sort** check box to allow this column to be sorted in the **Records** pane in the **Edit Job** window (**Find** a job in the **Job Dashboard** and either click **Edit** or right-click it and select **Edit**) or in the window that appears when a job is opened in the **Job Dashboard** (which appears when you select a record and either click **Open** or right-click it and select **Open**) You can select up to ten columns to be sorted.

   Note A column that is set as the record key is also a sortable column and count toward the ten columns that can be sortable.

   Sorting is based on the prefix of the input variable. If the prefix is **&**, sorting is based on timestamp (which in the end is a numeric sort); if the prefix is **$**, sorting is alphanumeric; and if the prefix is #, sorting is numeric.

   These values are all stored as strings regardless of the prefix the values are converted to when the user opens or edits a job to prepare a job. If, for example, the prefix is # and the value does not convert to a numeric value, the value is managed as *null*; for #ZIP and value SEVEN, seven is *null* when sorted.

   7. In the **Tooltip** field, type the text that appears when you hover your mouse over the column in the pane that displays records that appear when you open a job.
   8. Click **OK**.
6. To prevent duplicate records when this header is used to run jobs, you can set a column as the record key. Jobs run more quickly when a column is set as a record key.

   Selecting a record key allows Perceptive DataTransfer to process any records with different values in the selected column at the same time. You do not need to select a column that guarantees unique values, but the column should capture duplicate records. For example, if you select *Last Name* as the key to a header and process a file that has many records with the last name *Smith*, which represent

different students, you prevent any two records with the same last name form being processed at the same time.

To mark a column as the record key, select the column and click . The icon appears next to the column to denote that it is the record key. You can mark one column only as the record key. You cannot configure composite keys.

Note A column that is set as the record key is also a sortable column and counts toward the ten elements on which you can sort.

7. To delete a variable, select it and either right-click it and select **Delete** or click .
8. To preview data, in the **Data Preview** pane, click .
9. In the **Upload Preview File** dialog box, click **Browse** and navigate to the file that you want to preview, and then select it.
10. Click **Preview** to view data in the **Data Preview** pane.

    Note You can import data from the first sheet of a workbook only, and the data must be uniform; you cannot import data from charts that have dissimilar data.
11. To clear data from the pane, click .

Configuring and modifying XML headers and header variables

Note You do not provide input variables for XML header jobs, because XML syntax for procedures allows you to reference every node in the XML by the exact XML tag name. Other input files have an input variable for each data field in the file, but for XML input files, you must use the *for each* and *node* syntax to navigate to the desired part of the XML tree.

To configure an XML header, complete the following steps.

1. Select **Configuration > Headers**.
2. In the **Header Administration** dialog box, select the XML header you want to modify and click .
3. In the **Header Editor** dialog box, in the **Header Information** pane, complete the following substeps.
    1. In the **Header Information** pane, type the name of the header in the **Header Name** field.
    2. From the **Match Result Display** drop-down box, select the procedure that defines the information that displays in the **Results** pane in the **Job** window when a match occurs. Refer to Configuring Procedures for more information about procedures.

       For an overview of the steps required to configure jobs that use this query to populate the **Results** pane in the **Job** window, refer to Specifying data displayed in Results pane.
3. Perform one of the following steps.
    - If you are using an XSD document, proceed to step 5.
    - If you are using an XML document, proceed to step 6.
4. If you are using an XSDdocument to create the header, perform the following action.
    1. Place your main XSD file and any XSD schema files that are referenced within the main XSD file into a ZIP file.

       For example, for an XSD file with the name *HighSchoolTranscript_v1.1.0.xsd* that contains the following:

```
<?xml version="1.0" encoding="utf-8"?>

<xs:schema xmlns:xs="http://www.w3.org/2001/XMLSchema"
xmlns:core="core:CoreMain:v1.8.0" xmlns:AcRec="sector:AcademicRecord:v1.5.0"
xmlns:HSTrn="message:HighSchoolTranscript:v1.1.0"
targetNamespace="message:HighSchoolTranscript:v1.1.0" version="v1.1.0">

<xs:import namespace="core:CoreMain:v1.8.0"
schemaLocation="CoreMain_v1.8.0.xsd"/>

   <xs:import namespace="sector:AcademicRecord:v1.5.0"
schemaLocation="AcademicRecord_v1.5.0.xsd"/>

      <xs:element name="HighSchoolTranscript">

      <xs:complexType>

      <xs:sequence>

      <xs:element name="TransmissionData" type="AcRec:TransmissionDataType"/>

      <xs:element name="Student" type="AcRec:K12StudentType"/>

      <xs:element name="NoteMessage" type="core:NoteMessageType"
minOccurs="0"  maxOccurs="unbounded"/>

      <xs:element name="UserDefinedExtensions"
type="core:UserDefinedExtensionsType" minOccurs="0"/>

  </xs:element>

</xs:schema>
```

The file contains references to these schemas:

- <xs:import namespace="urn:org:pesc:core:CoreMain:v1.8.0" schemaLocation="CoreMain_v1.8.0.xsd"/>
- <xs:import namespace="urn:org:pesc:sector:AcademicRecord:v1.5.0" schemaLocation="AcademicRecord_v1.5.0.xsd"/>

HighSchoolTranscript_v1.1.0.xsd (the main file) and CoreMain_v_1.8.0.xsd and

AcademicRecord_v1.5.0.xsd (the referenced files) are placed in the ZIP file.

Note If a file that is referenced from the main file has references to other files, you must include those files in the ZIP file. For example, if AcademicRecord_v.1.5.0.xsd referenced other files, those must also be included in the ZIP file.

2. Obtain the name of your main XSD file, including the extension.

   In the preceding example, the main XSD file is *HighSchoolTranscript_v1.1.0.xsd.*

3. Open the XSD file and obtain the name of the root element of your document. In the preceding example, the root element is *HighSchoolTranscript.*

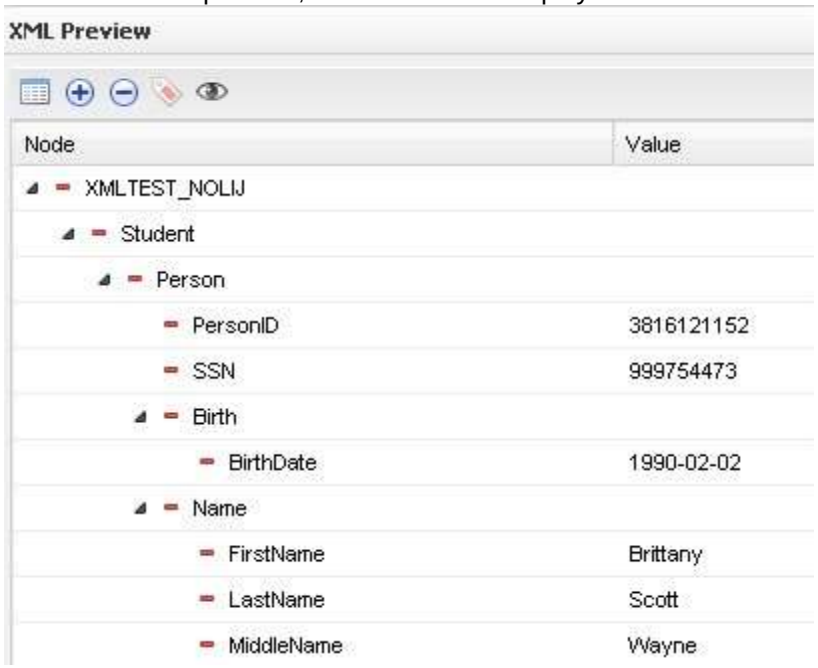
4. Preview data by completing the following substeps. Note that (you cannot preview data in tablet view).

   1. In the **XML Preview** pane, click  and select **Preview XSD File**. The U**pload XSD Preview File** dialog box appears.
   2. In the **Upload Zip File** field, click **Browse...**, navigate to the location of the ZIP file with the XSD file you want to preview, and select it.
   3. In the **Master Filename** field, type the name of the main XSD file. In the preceding example, the main XSD file is *HighSchoolTranscript_v1.1.0.xsd.*

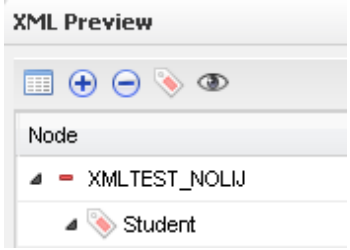
4. In the **Root Element** field, type the name of the root element. In the preceding example, the root element is *HighSchoolTranscript.*
5. Click **Preview**. The XSD file is uploaded, and its contents display in the **XML Preview** pane.

5. Proceed to step 7.
6. If you are using an XML document to create the header, preview data by completing the following substeps. Note that you cannot preview data in tablet view.
   1. In the **XML Preview** pane, click and select **Preview XML File**.
   2. In the **Upload XML Preview File** dialog box, click **Browse**, navigate to the location of the XML file you want to preview, and select it.
   3. Click **Preview**.

      The XML file is uploaded, and its contents display in the **XML Preview** pane.



7. To expand all notes in the XML tree, click . To collapse them, click .
8. Set a record identifier by selecting the appropriate XML node and clicking . The node set as the record identifier is the node that denotes the start of each record.

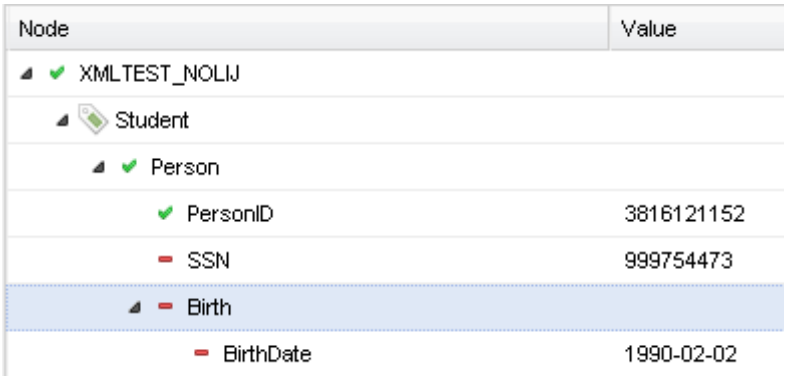


9. The XML tree refreshes and marks the corresponding nodes as record identifiers. The Identifier field in the Header Information pane displays the name of the identifier.

   Note: If you change the record identifier after you create a job using the XML header, the job will not run properly.

10. If you select an end name in the XML tree,  appears. If you click the icon, the XML tag displays in the **Records** pane (the topmost, leftmost pane in either the **Edit Job** or **Open Job** windows). You can also right-click a node and select **Set to Display**.

    A ✔ icon appears next to XML tags that appear in the **Records** pane. Nodes that are selected to appear in the **Records** pane are displayed in the **Element Order** pane.



    If you select a node that is marked to appear in the **Records** pane, the  appears. If you click this icon, the XML node will not be displayed in the **Records** pane. You can also right-click a node and select **Set to Hide**.

    A ▬ icon appears next to XML tags that do not appear in the **Records** pane.

11. Configure the order of XML elements specified to be displayed in the **Records** pane. The **Element Order** pane displays the nodes you selected for display.

    To configure the element order, select the appropriate node in the **Element Order** pane and complete one of the following actions.

    - Click  to move the node to be the first node to be displayed.
    - Click  to move the node up one position.
    - Click  to move the node down one position.
    - Click  to move the node to the last position.

12. To allow an element to be sortable in the **Records** panes in the **Edit Job** window (which appears when you select a job in the Job Dashboard and either click **Edit** or right-click it and select Edit) or in the window that appears when a job is opened in the Job Dashboard (which appears when you select a record and either click **Open** or right-click it and select **Open**), click the element and click  . You can select up to ten elements to be sortable.

    To remove sorting from the element, select an element on which sorting is enabled and click  . .

    Note Neither icon appears if you have selected the maximum of ten elements to be sortable and you select an element that is not sortable.

    Note An element that is set as the record key is also a sortable column and counts toward the ten columns that can be sortable.

13. To prevent duplicate records when this header is used to run jobs, you can set

14. Selecting a record key allows Perceptive DataTransfer to process any records with different values in the selected column at the same time. You do not need to select a column that guarantees unique values, but the column should capture duplicate records. For example, if you select Last Name as the key to a header and process a file that has many records with the last name Smith, which represent different students, you prevent any two records with the same last name from being processed at the same time.

    To mark a column as the record key, select the column and click  . The  icon appears next to the column to denote that it is the record key. You can mark one column only as the record key. You cannot configure composite keys.

    Note A column that is set as the record key is also a sortable column and counts toward the ten columns that can be sortable.

14. To hide a displayed node that appears in the **Element Order** pane, select the node and click  . The node is hidden and is removed from the **Element Order** pane.

15. To change the name of an XML node to display in the **Records** pane, complete the following substeps.
    1. Click the node you want to rename in the **Element Order** pane and click  .
    2. In the **Edit Display Name** dialog box, in the **Display Name** field, type the name to be displayed in the **Records** pane. The heading of the column is changed to this name when the XML node displays in the **Records** pane.
    3. Click **OK**.

       Note Changing the XML node name does not change the XML tag; it only changes the display name used in the **Records** pane when jobs are opened or edited.

       When you open or edit a job that uses an XML header, the columns in the **Records** pane are populated and ordered based on the tags you specified to be displayed and the order in which you displayed.

       You can view the full XML record in the **Records** pane; refer to Modifying records.

Configuring and modifying EDI headers and header variables

To configure an EDI header, complete the following steps.

Note: EDI headers are supported in Perceptive DataTransfer 6.4.4 and higher.

1. Select **Configuration > Headers**.
2. In the **Header Administration** dialog box, click the EDI header you want to modify and click  .
3. In the **Header Editor** dialog box, in the **Header Information** pane, complete the following substeps.
    1. Type the name of the header in the **Header Name** box.
    2. From the **Match Result Display** list, select the procedure that defines the information that displays in the **Results** pane in the **Job** window when a match occurs. Refer to Configuring Procedures for more information about procedures.

       For an overview of the steps required to configure jobs that use this query to populate the **Results** pane in the **Job** window, refer to Specifying data displayed in Results pane.
    3. From the EDI Format list, select the EDI format to use for the header. Refer to Configuring EDI formats for more information.

4. Preview data by completing the following substeps. Note that you cannot preview data in tablet view.
   1. In the rightmost pane, click and select **Preview EDI File**.
   2. Click **Browse**, navigate to the location of the EDI file you want to preview, and select it.
   3. Click **Preview**. The EDI file is uploaded, and its contents display in the pane in XML format.
5. To expand all nodes in the XML tree, click . To collapse them, click .
6. Set a record identifier by selecting the appropriate XML node and clicking . The node set as the record identifier is the node that denotes the start of each record.

   The XML tree refreshes and marks the corresponding nodes as record identifiers. The Identifier field in the Header Information pane displays the name of the identifier.

   Note If you change the record identifier after you create a job using the XML header, the job will not run properly.
7. If you select an end node in the XML tree, the appears. If you click the icon, the XML tag displays in the **Records** pane (the topmost, leftmost pane in either the **Edit Job** or **Open Job** windows). You can also right-click a node and select **Set to Display**.

   A icon appears next to XML tags that display in the **Records** pane.

   Nodes that are selected to appear in the **Records** pane are displayed in the **Element Order** pane.

   If you select a node that is marked to appear in the **Records** pane, the appears. If you click this icon, the XML node does not display in the **Records** pane. You can also right-click a node and select **Set to Hide**.

   A icon appears next to XML tags that do not appear in the **Records** pane.
8. Configure the order of XML elements specified to display in the **Records** pane. The **Element Order** pane displays the nodes you selected to display. To configure the element order, select the appropriate node in the **Element Order** pane and complete one of the following steps.

   - Click to move the node to be the first node to be displayed.
   - Click to move the node up one position.
   - Click to move the node down one position.
   - Click to move the node to the last position.
9. To allow an element to be sortable in the **Records** pane in the **Edit Job** window (which appears when you select a job in the **Job Dashboard** and either click **Edit** or right-click it and select **Edit**) or in the window that appears when a job is opened in the **Job Dashboard** (which appears when you select a record and either click **Open** or right-click it and select **Open**), click the element and click the . You can select up to ten elements to be sortable.

   To remove sorting from the element, select an element on which sorting is enabled and click .

   Note Neither icon appears if you have selected the maximum of ten elements to be sortable and you select an element that is not sortable.

   Note An element that is set as the record key is also a sortable column and counts toward the ten columns that can be sortable.

10. To prevent duplicate records when this header is used to run jobs, you can set a column as the record key. Jobs run more quickly when a column is set as a record key.

    Selecting a record key allows Perceptive DataTransfer to process any records with different values in the selected column at the same time. You do not need to select a column that guarantees unique values, but the column should capture duplicate records. For example, if you select *Last Name* as the key to a header and process a file that has many records with the last name *Smith*, which represent different students, you prevent any two records with the same last name from being processed at the same time.

    To mark a column as the record key, select the column and click  . The  icon appears next to the column to denote that it is the record key. You can mark one column only as the record key. You cannot configure composite keys.

    Note A column that is set as the record key is also a sortable column and counts toward the ten columns that can be sortable.

11. To hide a displayed node that appears in the **Element Order** pane, select the node and click  . The node is hidden and is removed from the **Element Order** pane.

12. To change the name of an XML node to display in the **Records** pane, complete the following actions.

    1. Click the node you want to rename in the **Element Order** pane and click  .
    2. In the **Edit Display Name** dialog box, in the **Display Name** field, type the name to display in the **Records** pane. The heading of the column is changed to this name when the XML node displays in the **Records** pane.
    3. Click **OK**.

       Note Changing the XML node name does not change the XML tag; it only changes the display name used in the **Records** pane when jobs are opened or edited.

       When you open or edit a job that uses an XML header, the columns in the **Records** pane are populated and ordered based on the tags you specified to be displayed and the order in which you displayed.

       You can view the full XML record in the **Records** pane; refer to Modifying records.

Configuring Hobsons Connect CRM headers and header variables

To configure Hobsons Connect CRM header, complete the following steps.

1. Select **Configuration > Hea**ders.
2. In the **Header Administration** dialog box, click the **Hobsons Connect CRM** header you want to modify and click .

3. In the **Header Editor** dialog box, in the **Header Information** pane, complete the following substeps.
    1. In the **Header Name** field, type the name of the header.
    2. From the **Match Result Display** drop-down box, select the procedure that defines the information that displays in the **Results** pane in the **Job** window when a match occurs. Refer to Configuring Procedures for more information about procedures.

       For an overview of the steps required to configure jobs that use this query to populate the Results pane in the **Job** window, refer to Specifying data displayed in Results pane.
    3. From the **Connection Group** drop-down box, select the connection group to use for the header. The connection group must have a Hobsons connection configured; the header uses the Hobsons connection, from the selected connection group.

       Note The connection is not saved on the header; you must select the connection each time you modify the header.

       The **Header Filters pane** appears after you select a connection group.
4. To add a Hobsons Connect CRM filter, in the **Header Filters** pane, complete the following substeps.
    1. Click  .
    2. In the **Select Filters** dialog box, select one filter to add or select multiple filters by performing one of the following actions. Note that the dialog box displays only those filters that you have selected from the Hobsons connection.
        - Click a filter and press and hold the **CTRL** key as you click each filter with your mouse.
        - Click a filter and press and hold the **SHIFT** key as you click another filter to select contiguous filters.
    3. Click **OK**.
5. To delete a filter, in the **Header Filters** pane, select the filter you want to delete and click .
6. To add a new variable, perform the following substeps.
    1. Click  in the **Header Variables** pane. The variables are retrieved from the Hobsons connection and display in a new dialog box.
        - If you do not click a variable in the table, the new variable appears as the first entry in the table.
        - If you click a variable, the new variable is added below the selected variable.
    2. Select one variable to add or select multiple variables by performing one of the following actions.
        - Click a variable and press and hold the **CTRL** key as you click each variable with your mouse.
        - Click a variable and press and hold the **SHIFT** key as you click another variable to select contiguous variables.
    3. Click **OK**.
7. To modify a header variable, in the **Header Variables** pane, complete the following substeps.
    1. Select the variable you want to modify and either right-click it and select **Edit** or  .
    2. In the **Edit Input Variable** dialog box, in the **Position** field, type the number of the position of the column. You may need to adjust the column position if a data file changes.
    3. In the **Name** field, type the name of the variable.

4. From the **Data Type** drop-down box, select the data type of the variable (*Character*, *Number*, or *Date*).

   In the **Name** field, the appropriate symbol used to specify the data type of the variable is appended to the first character of the name.

   The **$** (dollar sign) denotes a character variable, # (number sign) denotes a numeric variable, and **&** (ampersand) denotes a date variable.

5. Select the **Display** check box to display the column in the pane that displays records in the window that appears when you open a job.
6. Select the **Sort** check box to allow this column to be sorted in the **Records** pane in the **Edit Job** window (**Find** a job in the **Job Dashboard** and either click **Edit** or right-click it and select **Edit**) or in the window that appears when a job is opened in the **Job Dashboard** (which appears when you select a record and either click **Open** or right-click it and select **Open**) You can select up to ten columns to be sorted.

   Note A column that is set as the record key is also a sortable column and count toward the ten columns that can be sortable.

   Sorting is based on the prefix of the input variable. If the prefix is **&**, sorting is based on timestamp (which in the end is a numeric sort); if the prefix is **$**, sorting is alphanumeric; and if the prefix is #, sorting is numeric.

   These values are all stored as strings regardless of the prefix the values are converted to when the user opens or edits a job to prepare a job. If, for example, the prefix is # and the value does not convert to a numeric value, the value is managed as *null*; for #ZIP and value SEVEN, seven is *null* when sorted.

7. Click **OK**.

8. To prevent duplicate records when this header is used to run jobs, you can set a column as the record key. Jobs typically run more quickly when a column is set as a record key.

   Selecting a record key allows Perceptive DataTransfer to process any records with different values in the selected column at the same time. You do not need to select a column that guarantees unique values, but the column should capture duplicate records. For example, if you select *Last Name* as the key to a header and process a file that has many records with the last name *Smith*, which represent different students, you prevent any two records with the same last name from being processed at the same time.

   To mark a column as the record key, select the column and click  . The  icon appears next to the column to denote that it is the record key. You can mark one column only as the record key. You cannot configure composite keys.

   Note A column that is set as the record key is also a sortable column and count toward the ten columns that can be sortable.

9. To delete a variable, in the **Header Variables** pane, select the variable you want to delete and click  .

10. To preview data in the **Data Preview** pane, click  . You cannot preview data in tablet view.

11. Perceptive DataTransfer runs the Hobsons Connect CRM web service for the selected filter and input variables; the data returned from the web service displays in **Data Preview** pane.

12. To clear data from the pane, click  .

Configuring and modifying web service headers

Use a web service header when you are using web services or SOAP web services to retrieve data; refer to Configuring data providers for more information.

To configure a web service header, complete the following steps.

1. Select **Configuration > Headers**.
2. In the **Header Administration** dialog box, click the web service header you want to modify and either right-click it and select **Edit** or click  .
3. In the **Header Editor** dialog box, in the **Header Information** pane, complete the following substeps.
    1. Type the name of the header in the **Header Name** field.
    2. From the **Match Result Display** drop-down box, select the procedure that defines the information that displays in the **Results** pane in the **Job** window when a match occurs. Refer to Configuring Procedures for more information about procedures.

       For an overview of the steps required to configure jobs that use this query to populate the **Results** pane in the **Job** window, refer to Specifying data displayed in Results pane.
4. In the **Web Service Properties** pane, complete the following substeps.
    1. From the **Web Service** drop-down box, select the web service data provider with which you want to configure the header.
    2. Select the connection group associated with the web service you want to use from the **Connection Group** drop-down box. This drop-down box displays the connection groups that contain web service connections that match the name of the connection selected when you configured the web service data provider.
5. Click the **Capture** button to create the header input variables using the output parameters you configured for the data provider.
6. To prevent duplicate records when this header is used to run jobs, you can set a column as the record key. Jobs run more quickly when a column is set as a record key.

   Selecting a record key allows Perceptive DataTransfer to process any records with different values in the selected column at the same time. You do not need to select a column that guarantees unique values, but the column should capture duplicate records. For example, if you select *Last Name* as the key to a header and process a file that has many records with the last name *Smith*, which represent different students, you prevent any two records with the same last name from being processed at the same time.

   To mark a column as the record key, select the column and click  . The  icon appears next to the column to denote that it is the record key. You can mark one column only as the record key. You cannot configure composite keys.

   Note A column that is set as the record key is also a sortable column and counts toward the ten elements on which you can sort.
7. To delete a variable in the **Header Variables** pane, select it and click  .
8. To preview data, in the **Data Preview** pane, click  .
9. In the **Upload Preview File** dialog box, click **Preview** to view data in the **Data Preview** pane.

10. To clear data from the pane, click .

Removing sharing on headers

To remove sharing on a header that is shared with another workspace, complete the following steps.

1. Select **Configuration > Headers**.
2. In the **Header Administration** dialog box, click the header and click .

   Sharing on the header is removed.

Rolling back headers

If changes have been made and committed to a header, you can use a previous version (*roll back*) of it.

1. Select **Configuration > Headers**.
2. In the **Header Administration** dialog box, select the header you want to roll back and either right-click it and select **Rollback Header** or click .

   The **Rollback Manager** dialog box appears and displays the following information about previous versions of the procedure.

   - Name: Name of the header.
   - Status: Status of the header (*current* or *dated*).
   - Version Date: Date on which the version was created.
   - Created By: User name of the user who created the version.
   - Comments: Comments that were added about the version.
3. Select the version to which you want to revert and click **Rollback**.

   The header is updated to use the selected previous version of the header.

Obtaining the latest versions of headers

If changes have been made to and committed to a header, you can obtain the latest version of it.

1. Select **Configuration > Headers**.
2. In the **Header Administration** dialog box, select the header for which you want to obtain the latest version and click .

   The header is updated to the latest version.

Deleting headers

To delete a header, complete the following steps.

1. Select **Configuration > Headers**.
2. In the **Header Administration** dialog box, select the header that you want to delete and click . A message appears, prompting you to confirm that you want to delete the header.
3. Click **Yes**.

   The header is deleted and is removed from the **Header Administration** dialog box.

Exporting headers

Note You cannot export headers in tablet view.

To export headers to an XML file, complete the following steps.

1. Select **Configuration > Headers**.
2. In the **Header Administration** dialog box, select the header you want to export.
3. Click  and select **Export**.
4. Save the file to the appropriate location.

Importing headers

Note You cannot import headers in tablet view.

You can import headers from an XML file by completing the following steps.

1. Select **Configuration > Headers**.
2. In the **Header Administration** dialog box, click  and select **Import**.
3. In the **Import File Source** dialog box, click the **Browse** button and navigate to the file that you want to import or type the location of the file in the **File** field.
4. Select the **Update Existing Entities** check box to update an existing header, with the same name as the header you are importing, with the header you are importing.

   If you do not select this check box, and there is an existing header with the same name as the header you are importing, *DUPLICATE-* is appended to the start of the name of the header you are importing; the existing header is not updated. If a header exists with the name DUPLICATE-<*header_name*>, *DUPLICATE(2)-* is appended to the start of the name. The number continues to be incremented so that all headers have unique names.
5. Click **OK**.

Configuring Forms

You can configure forms to use to compare the current input record side-by-side with the possible existing record. You can link fields in the form to fields in the record and compare the values to view whether they match.

One form is contained in a workspace and is used for all jobs run within that workspace.

You can share a form with other workspaces, and you can import a form from and export a form to an XML file.

Note The form is displayed for users in the Job window only when a job is running in interactive mode.

Note You cannot configure forms in tablet view.

To open the form editor, select **Configuration > Forms**. You can also configure forms by selecting the appropriate job in the Job Dashboard, right-clicking it, and then selecting **Edit**.

When users run records, they can use the form to verify that information in these linked fields matches. Refer to the following topics for more information.

Refer to the following topics for more information.

- Understanding Form Objects
- Adding Forms
- Adding Form Objects
- Configuring Form and Form Object Properties
- Linking Form Fields and Assigning Variables to Form Fields
- Using Form Toggles
- Moving Form Objects
- Aligning Form Objects
- Copying and Pasting Form Objects
- Configuring Form Colors
- Deleting Form Objects
- Removing Sharing on Shared Forms
- Rolling Back Forms
- Obtaining the Latest Version of Forms

Understanding Form Objects

The following table provides a description of the form objects you can add to a form.

Object	Description
Field Set	Area that contains a group of related information.
Label	Text that describes an area on the form.
TextField	Field in which data can be displayed.
Check box	Element that users can check to select an option.

Adding Forms

You can create a new form from a blank template, or you can add a form that is shared by another workspace. You can also create a new form from a copy of an existing form.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The Form Editor appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Select **Tools > Add Form**.
3. In the **Add Verification Form** dialog box, perform one of the following actions.
   - To add a new form, click the **Add Verification Form** tab. Next, perform one of the following actions.
     - Click the **From Blank** radio button to create a new form based on a blank template.
     - Click the **From Copy** radio button if you have a form configured; you create new form based on a copy of this existing form.
   - To add a form that is shared by another workspace, click the **New Shared Verification Form** tab; then, from the Workspace drop-down box, select the workspace that shares the form you want to add.
4. Click **OK**. One of the following actions occurs.
   - If you added a blank form, the Form Editor is empty, and you can begin creating the form.
   - If you added a form as a copy from a selected form, a copy of the form appears in the Form Editor.
   - If you added a shared form, the shared form appears in the Form Editor.

Adding Form Objects

You can add labels, text fields, check boxes, and field sets to the background of the form. You can add labels, text fields, and check boxes to field sets.

To add a new form object, complete the following steps.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The Form Editor appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. To add a field set, either right-click the background of the form and select **Add > Field Set** or select **Add > Field Set** from the menu at the bottom of the window. The field set appears in the form.
3. To add an object, do the following.
   - To add an object to a field set, right-click the field set, select **Add**, and then select the appropriate object (*Label*, *TextField*, or *Check box*); or, click **Add** in the menu at the bottom of the window and select the appropriate object. The object appears in the field set.
   - To add an object to the background of the form, right-click the form, select **Add**, and then select the appropriate object (*Label*, *TextField*, or *Check box*); or, click **Add** in the menu at the bottom of the window and select the appropriate object. The object appears in the form.
4. Click **Save** to save your changes.

Configuring Form and Form Object Properties

Refer to the following topics for more information.

- Configuring Default Form Properties
- Configuring Field Set Properties
- Configuring Text Field Properties
- Configuring Check Box Properties

Configuring Default Form Properties

To configure default form properties, complete the following steps.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The Form Editor appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Right-click the form background and select **Properties** from the popup menu.
3. In the **Properties** dialog box, complete the following steps.
4. In the **Field Set Height** field, type the height, in pixels, of the field sets.
5. In the **Field Set Width** field, type the width, in pixels, of the field sets.
6. In the **Component Width** field, type the width, in pixels, of the form components.

7. In the **Label Width** field, type the width, in pixels, of the form labels.
8. In the **Alignment** field, click the appropriate radio button to configure the alignment of form objects.
   - Left: Aligns all objects to the left, aligning the left edges of all objects to the left edge of the form.
   - Right: Aligns all objects to the right, aligning the right edges of all objects to the right edge of the form.
   - Top: Aligns all objects to the top, aligning the top edges of all objects to the top edge of the form.
9. Click **OK**.

These properties are applied, by default, to all objects you add to the form.

Configuring Field Set Properties

To configure field set properties, complete the following steps.

1. Perform one of the following actions.
   - Select **Configuration > Forms** to display the Form Editor.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Right-click the field set and select **Properties** from the popup menu.
3. In the **Properties** dialog, in the **Base Info** pane, provide the appropriate information in the following fields.
   - ID: Type the ID that uniquely identifies the field set. You can also type this value in the ID field in the bottom of the Form Editor.
   - Order: Type the order value of the field set, which is either *zero* (*0*), the higher value, or *1*. The order determines which object appears on top of another object if they overlap. An object with a 0 value appears above an object with a 1 value.
   - Top: Type the value, in pixels, of the top margin. This value specifies boundary length between the top of the verification form and the field set. You can also type this value in the Top field in the bottom of the Form Editor.
   - Left: Type the value, in pixels, of the left margin. This value specifies the boundary length between the left side of the verification form and the field set. You can also type this value in the Left field in the bottom of the Form Editor.
   - Height: Type the field set height, in pixels. You can also type this value in the Height field in the bottom of the Form Editor.
   - Width: Type the field set width, in pixels. You can also type this value in the Width field in the bottom of the Form Editor.
4. In the **Fields Properties** pane, specify the appropriate information.
   - Heading: Type the name of the heading, which is the text that appears in the top of the field set to identify it. You can also type this name in the Heading field in the bottom of the Form Editor.
   - User Check box: Click the appropriate radio button to enable or disable field toggles for users. Field set toggles are used to determine what information, if any, can be uploaded to the database when differences in linked form fields are detected. Refer to Using Form Toggles for more information.

5. Click **OK** to apply your changes and close the dialog box.
6. Click **Save** to save your changes.

Configuring Text Field Properties

To configure text field properties, complete the following steps.

1. Perform one of the following actions.
   - Select **Configuration > Forms** to display the Form Editor.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Right-click the text field and select **Properties** from the popup menu.
3. In the **Properties** dialog box, in the **Base Info** pane, provide the appropriate information in the following fields.
   - ID: Type the unique ID that identifies the text field. You can also type this value in the ID field in the bottom of the Form Editor.
   - Order: Type the order value of the text field, which is either *zero* (*0*), the higher value, or *1*. The order determines which object appears on top of another object if they overlap. An object with a 0 value appears above an object with a 1 value.
   - Top: Type the value, in pixels, of the top margin. This value specifies boundary length between the top of the text field and the field set. You can also type this value in the **Top** field in the bottom of the Form Editor.
   - Left: Type the value, in pixels, of the left margin. This value specifies the boundary length between the left side of the text field and the field set. You can also type this value in the **Left** field in the bottom of the Form Editor.
   - Height: Type the text field height, in pixels. You can also type this value in the **Height** field in the bottom of the Form Editor.
   - Width: Type the text field width, in pixels. You can also type this value in the **Width** field in the bottom of the Form Editor.
4. The **Advanced Properties** displays information that appears in the form field and information linking form fields together. Refer to Linking Form Fields and Assigning Variables to Form Fields for more information.
5. In the **Field Properties** pane, specify the appropriate information:
   - Label: Type the text that appears in the form to identify the text field.
   - Label Width: Type the width, in pixels, of the text field.
   - Field Width: Type the width, in pixels, of the text field.
   - Alignment: Click the appropriate radio button.
     - Left: Specifies that the label appears to the left of the text field.
     - Top: Specifies that the label appears above the text field.
     - Right: Specifies that the label appears to the right of the text field.
6. Click **OK** to apply your changes and close the dialog box.

7. To make the text field read-only so that users cannot modify information in the field when they view the verification form, check the **Read Only** check box at the bottom of the Form Editor.
8. Click **Save** to save your changes.

Configuring Check Box Properties

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The Form Editor appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Right-click the check box and select **Properties** from the popup menu.
3. In the **Properties** dialog box, in the **Base Info** pane, provide the appropriate information in the following fields.
   - ID: Type the ID that uniquely identifies the check box. You can also type this value in the **ID** field in the bottom of the Form Editor.
   - Order: Type the order value of the check box, which is either *zero* (*0*), the higher value, or *1*. The order determines which object appears on top of another object if they overlap. An object with a 0 value appears above an object with a 1 value.
   - Top: Type the value, in pixels, of the top margin. This value specifies boundary length between the top of the check box and the field set. You can also type this value in the **Top** field in the bottom of the Form Editor.
   - Left: Type the value, in pixels, of the left margin. This value specifies the boundary length between the left side of the check box and the field set. You can also type this value in the **Left** field in the bottom of the Form Editor.
   - Height: Type the check box height, in pixels. You can also type this value in the **Height** field in the bottom of the Form Editor.
   - Width: Type the check box width, in pixels. You can also type this value in the **Width** field in the bottom of the Form Editor.
4. In the **Field Properties** pane, specify the appropriate information:
   - Label: Type the text that appears in the form to identify the check box.
   - Label Width: Type the width, in pixels, of the label.
   - Field Width: Type the width, in pixels, of the check box.
   - Alignment: Click the appropriate radio button.
     - Left: Specifies that the label appears to the left of the check box.
     - Top: Specifies that the label appears above the check box.
     - Right: Specifies that the label appears to the right of the check box.
5. Click **OK** to apply your changes and close the dialog box.
6. Click **Save** to save your changes.

Configuring Label Properties

To configure label properties, complete the following steps.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The Form Editor appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Click the label for which you want to configure properties.
3. In the **ID** field at the bottom of the form editor, type the ID that uniquely identifies the label.
4. In the **Top** field, type the value, in pixels, of the top margin. This value specifies boundary length between the top of the label and the field set.
5. In the **Left** field, type the value, in pixels, of the left margin. This value specifies the boundary length between the left side of the label and the field set.
6. In the **Width** field, type the label width, in pixels.
7. In the **Height** field, type the label width, in pixels.
8. Click **Save** to save your changes.

Linking Form Fields and Assigning Variables to Form Fields

You can configure text boxes to display information from the database by assigning it a program variable, which contains the appropriate information.

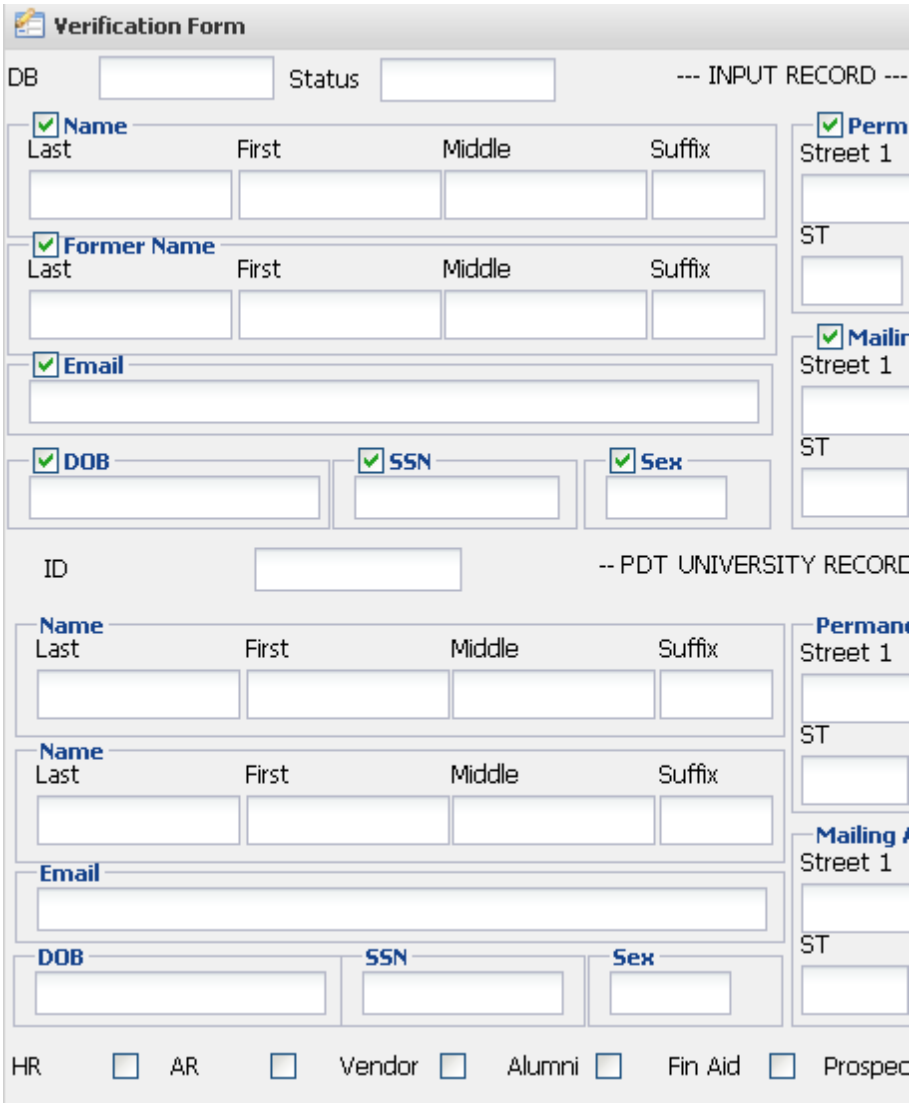
You can also link form fields to each other so that users can compare the input record to a potential match in the database. You link each field to a program variable, which returns the appropriate information from the database. Users can view whether the fields match in the verification form that is shown when they run interactive jobs.

To link form fields and to assign program variables to fields, do the following.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The Form Editor appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. To link form fields that have matching values, select the form fields by clicking on one form field and then pressing and holding the **Shift** key while clicking the matching field.

   When you view the **Properties** dialog box for a field, the Match To field in the **Advanced Properties** pane displays the ID of the other form field that is linked to the field.
3. Specify the variable to which the text field is assigned by selecting the variable from the **Program Variables** pane and dragging and dropping it onto the text field. For more information about program variables, refer to Configuring Procedures.

   When you view the **Properties** dialog box for the field, the Assigned To field in the **Advanced Properties** pane displays the program variable assigned to the field.



For example, in the preceding figure, the DOB fields for the input record (Input Record) and the database record (University Record) are linked.

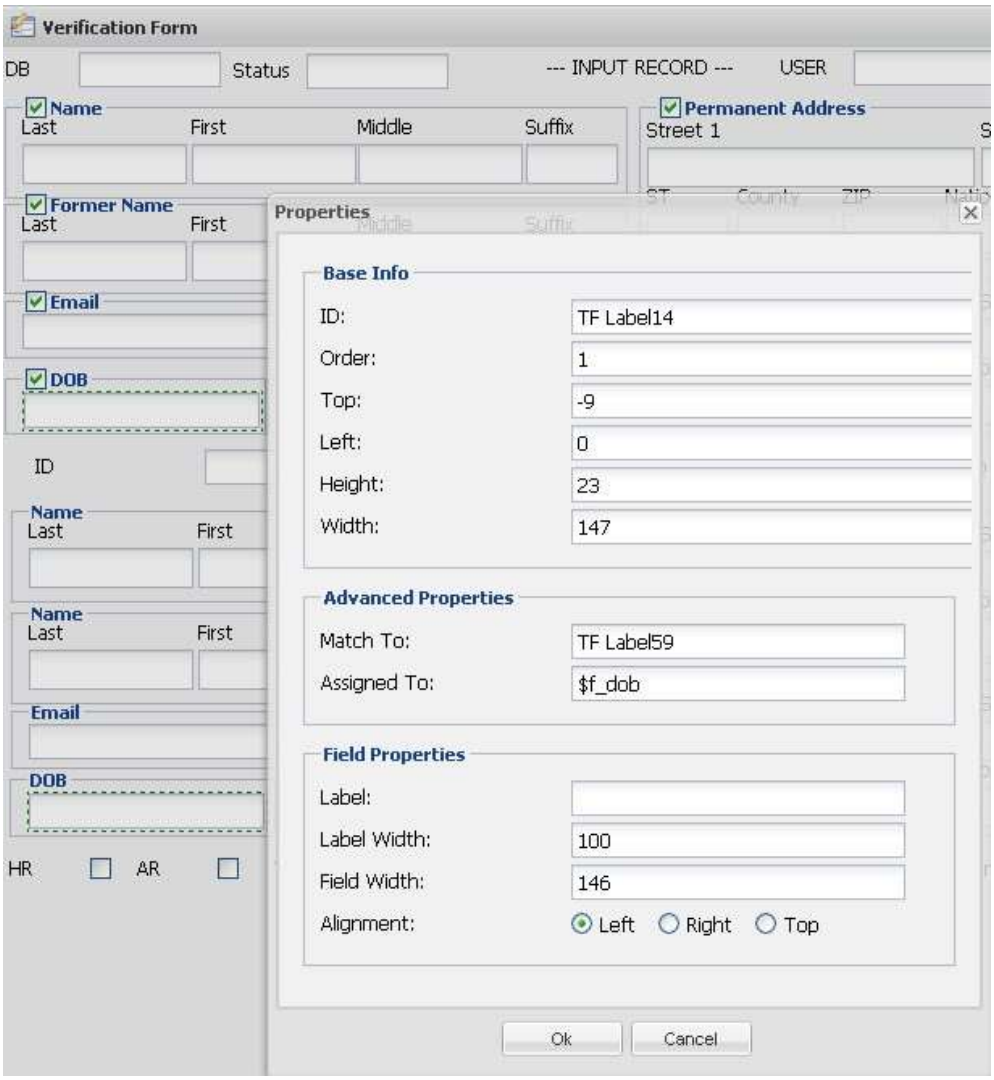
- The Input Record DOB field has an ID of *TF Label 14*.
- The University DOB field has an ID of *TF Label 59*.
- The Input Record DOB field is assigned to the *$f_dob* variable.
- The University Record DOB field is assigned to the *$d_dob* variable.

To link the fields, you would select the **Input Record DOB** field and then press and hold the **Shift** key on your keyboard while clicking the **University Record DOB** field.

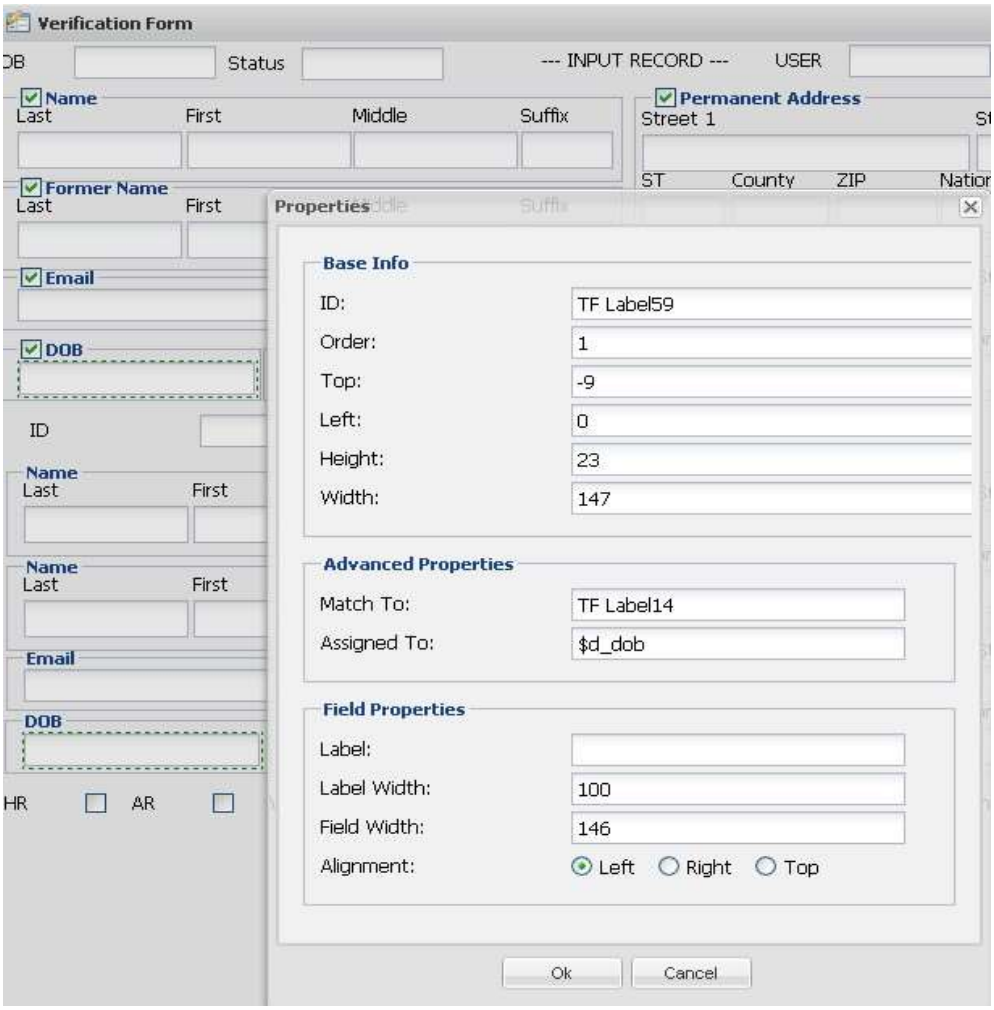
To assign the $f_dob variable to the Input Record DOB field, you would click the **$f_dob** variable in the Program Variables pane and then drag and drop it onto the text field to which it is assigned.

When you open the Properties dialog box for the Input Record DOB field, it displays the label (*TF Label*

59) to which it is linked and the variable (*$f_dob*) to which it is assigned.



When you selected the Input Record DOB and University Record DOB fields, you linked them both. To assign the $d_dob variable to the field, you would select the variable in the Program Variables pane and then drag and drop the variable onto the University Record field. When you open the Properties dialog box for the field, it displays the appropriate information for the field to which it is linked and the variable for which it is assigned.



4. Click **OK**.

   You can also select a color to highlight linked fields when their values differ from each other. By default, this color is a shade of blue; however, you can specify a color from a color palette. Refer to Configuring Form Match Colors.

Using Form Toggles

Refer to the following topics for more information.

- Understanding Form Toggles
- Configuring Form Toggles

Understanding Form Toggles

If a field set contains a field that is linked to another field, this check box is automatically enabled. If data between two field sets are the same (linked fields return the same values); the check box is unchecked; if the data is different, the toggle is checked.

Users can check or uncheck the toggle check boxes to specify which field sets of information are included when upload logic is processed.

Use form toggles if you need to provide the value of each field set to the associated upload logic; when the upload logic is run, this value is included when the data file is uploaded. The form can return the value of a toggle given a specified program variable name.

For more information about configuring procedures, refer to Configuring Procedures. Syntax for the toggle function is:

```
toggle(variable)
```

For example:

```
if toggle($suffix)
```

Syntax for the toggle_off function is:

```
toggle_off(variable)
```

For example:

```
toggle_off($last)
```

Syntax for the toggle_on function is:

```
toggle_on(variable)
```

For example:

```
toggle_on($last)
```

Configuring Form Toggles

To configure toggles for field sets, do the following.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The **Form Editor** appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Right-click the field set and select **Properties** from the popup menu.
3. In the **Properties** dialog box, in the **Field Properties** pane, select the appropriate option for the Use Check box option. Click the **Yes** radio button to use a check box (toggle) for the field set; click the **No** radio button if you do not want to use a check box.
4. Click **OK** to save your changes.

Moving Form Objects

You can move field sets and objects by dragging and dropping them with your mouse; complete the following steps.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The **Form Editor** appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Click the object that you want to move. To select multiple objects, click an object and press and hold the **Ctrl** key as you click each object with your mouse. You can deselect an object by clicking it and pressing and holding the **Ctrl** key; you can deselect all objects by clicking in an empty background area of the form.
3. With your mouse, drag and drop the object to the desired location.

The object is moved to the specified location. If you selected multiple objects, they are moved the same relative distance.

Aligning Form Objects

You can align form objects in a field set with each other, or you can align field sets on the form background with each other.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The **Form Editor** appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Select the objects you want to align. To select multiple objects, click an object and press and hold the **Ctrl** key as you click each object with your mouse. You can deselect an object by clicking it and pressing and holding the **Ctrl** key; you can deselect all objects by clicking in an empty background area of the form.
3. Right-click the objects and select **Align**.
4. Select one of the following.
   - Left: Aligns all objects to the left, aligning the left edges of all objects to the left edge of the first object you selected.
   - Right: Aligns all objects to the right, aligning the right edges of all objects to the right edge of the first object you selected.
   - Top: Aligns all objects to the top, aligning the top edges of all objects to the top edge of the first object you selected.
   - Bottom: Aligns all objects to the bottom, aligning the bottom edges of all objects to the bottom edge of the first object you selected.

Copying and Pasting Form Objects

You can copy and paste form objects located within a field set onto the same field set and copy and paste field sets onto the background form.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The Form Editor appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Select the objects you want to copy. To select multiple objects, click an object and press and hold the **Ctrl** key as you click each object with your mouse. You can deselect an object by clicking it and pressing and holding the **Ctrl** key; you can deselect all objects by clicking in an empty background area of the form.
3. Right-click the object and select **Copy**.
4. Perform one of the following actions.
   - To paste field sets onto the form background, right-click the background and select **Paste**.
   - To paste objects on a field set within the same field set, right-click the field set and click **Paste**.

Configuring Form Colors

Refer to the following topics for more information.

- Configuring Form Background Colors
- Configuring Field Set Foreground and Background Colors
- Configuring Label Foreground and Background Colors
- Configuring Check Box Foreground and Background Colors
- Configuring Text Field Foreground and Background Colors
- Configuring Form Match Colors

Configuring From Background Colors

To configure background colors for the form, do the following.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The **Form Editor** appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Right-click the background and select **Color**.
3. To set the form background color, select **Background**, and then select the color from the palette. The background of the form is set to this color.

Configuring Field Set Foreground and Background Colors

To configure background and foreground colors for a field set, do the following.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The **Form Editor** appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Right-click the field set and select **Color**.
3. To set the field set background color, select **Background**, and then select the color from the palette. The background of the field set is set to this color.
4. To set the field set foreground color, select **Foreground**, and then select the color from the palette.

If you have not configured colors for other form objects, any form objects that appear on the field set are set to this color. For example, if you have a label on the field set, the label is set to this color. Any additional form objects you add to the field set are also set to this color.

If you configure colors for form objects themselves, however, those colors, *not* the ones configured for the field set foreground or background, are used.

Configuring Label Foreground and Background Colors

To configure background and foreground colors for a label, do the following.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The **Form Editor** appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Right-click the label and select **Color**.
3. To set the label background color, select **Background**, and then select the color from the palette. The background of the label is set to this color.
4. To set the label foreground color, select **Foreground**, and then select the color from the palette. The color is applied to the label text of the label.

Configuring Check Box Foreground and Background Colors

To configure background and foreground colors for a check box, do the following.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The **Form Editor** appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Right-click the check box and select **Color**.

3. To set the check box background color, select **Background**, and then select the color from the palette. The background of the check box is set to this color.
4. To set the check box foreground color, select **Foreground**, and then select the color from the palette. The color is applied to the label text of the check box.

Configuring Text Field Foreground and Background Colors

To configure background and foreground colors for a text field, do the following.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The **Form Editor** appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Right-click the text field and select **Color**.
3. To set the text field background color, select **Background**, and then select the color from the palette. The background of the text field is set to this color.
4. To set the text field foreground color, select **Foreground**, and then select the color from the palette. The color is applied to the label text of the text field.

Configuring Form Match Colors

You can set the form match color, which is the color that highlights all linked form objects when they contain different values. For example, if the input record DOB and data record DOB fields are linked, and their values do not match, the text fields are displayed in the match color you specify. By default, the color is a shade of blue.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The **Form Editor** appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Right-click the background and select **Color**.
3. Select **Match**, then select a color from the color palette.

Deleting Form Objects

To delete a form object, do the following.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The **Form Editor** appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Select the object you want to delete and either right-click it and select **Delete** or click the **Delete** button.

   Note If you delete a field set, the field set and all of its objects are deleted. The object is deleted and is removed from the Form Editor.

13. Click **Save** to save your changes.

Removing Sharing on Shared Forms

To remove sharing on a form that is shared with another workspace, do the following.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The **Form Editor** appears.
   - In the Job Dashboard, select the job with the form you want to configure, then either right-click it and select **Edit** or click the **Edit** button.
2. Click **Tools** and select **Remove Sharing**. Sharing is removed on the form.

Rolling Back Forms

If changes have been made to and committed to a form, you can use a previous version *(roll back*) of it.

1. Perform one of the following actions.
   - Select **Configuration > Forms**. The **Form Editor** appears.
   - In the Job Dashboard, select the job with the form you want to roll back, then either right-click it and select **Edit** or click the **Edit** button.
2. Click **Tools** and select **Rollback**. The Rollback Manager dialog box appears and displays the following information.
   - Name: Name of the form.
   - Status: Status of the form (*pending*, *current*, or *dated*).
   - Version Date: Date and time on which the version of the form was created.
   - Created By: Username of the user who created the form.
   - Comments: Comments added by the user.
3. Select the version you want to use and click **Rollback**. The previous version is loaded into the Form Editor.

Obtaining the Latest Versions of Forms

If changes have been made to and committed to a form, you can obtain the latest version of it.

1. Preform one of the following actions.
   - Select **Configuration > Forms**. The **Form Editor** appears.
   - In the Job Dashboard, select the job with the form for which you want to obtain the latest version, and then either right-click it and select **Edit** or click the **Edit** button.
2. Click **Configuration** and select **Get Latest**. The latest version of the form is obtained and is loaded into the Form Editor.

Configuring Reports

You can write custom XML reports, which are based on JasperReports and are presented in PDF format. For more information about JasperReports, refer to http://en.wikipedia.org/wiki/JasperReports.

Note Reports only run against the Perceptive DataTransfer database. Refer to the following topics for more information.

- Configuring Reports.
- Configuring Scheduled Reports
- Viewing Reports

Configuring Reports

In Reports > Configure, you can add, modify, and delete reports; you can also import and export reports. Refer to the following topics for more information.

- Adding and Modifying Reports
- Deleting Reports
- Importing Reports
- Exporting Reports

Adding and Modifying Reports

You can create reports in JasperReports format.

1. Select **Reports > Configure**.
2. In the **Configure Reports** dialog box appears perform one of the following actions.
   - To add a new report, click  to display the **Add Report** dialog box.
   - To modify a report, select the report and either right-click it and select **Edit Report** or click  to display the **Edit Report** dialog box.
3. In the **Report Name** field, type the name of the report.
4. In the **XML** field, type the XML code that generates the report.
5. To add report parameters, complete the following substeps.
   1. In the **Report Parameters** pane, click  .
   2. In the **New Report Parameter** dialog box, from the **Parameter** drop-down box, select the type of parameter you want to add. Options are:
      - Header
      - Job
      - Start Date
      - End Date
      - User
   3. Click **Apply**.

4. Continue adding parameters, as appropriate, and click **OK** when you are finished.
5. To delete a parameter, select the parameter in the Report Parameters pane and click .

6. Click **OK** to save your changes and close the dialog box.

Deleting Reports

To delete a report, do the following.

1. Select **Reports > Configure**.
2. In the **Configure Reports** dialog box, select the report you want to delete and either right-click it and select **Delete Report** or click A dialog box appears, prompting you to confirm that you want to delete the report.
3. Click **Yes**. The report is deleted and is removed from the Configure Reports dialog box.

Importing Reports

To import reports from an XML file, do the following.

1. Select **Reports > Configure**.
2. In the **Configure Reports** dialog box, click and select **Import**.
3. In the **Import Report** dialog box, click **Browse**, navigate to the location of the file, and select it.
4. Click **OK**.

The file is imported, and reports that are imported are displayed in the Configure Reports dialog box.

Exporting Reports

To export reports to an XML file, do the following.

1. Select **Reports > Configure**.
2. In the Configure Reports dialog box, select the report you want to export.
3. Click and select **Export**.

Save the XML file in the appropriate location.

Configuring Scheduled Reports

In Reports > Schedule, you can add and delete reports that are scheduled to be run at specified times, and you can modify the schedule for these reports. You can also view reports by clicking (refer to Configuring Scheduled Reports for more information).

Refer to the following topics for more information.

- Adding and Modifying Scheduled Reports
- Deleting Scheduled Reports

Adding and Modifying Scheduled Reports

To add or modify a scheduled report, complete the following steps.

1. Select **Reports > Schedule**.
2. In the **Schedule Reports** dialog box, perform one of the following actions.
   - To add a new report, click  to display the **Add Scheduled Report** dialog box.
   - To modify a report, select the report and either right-click it and select **Edit Report** or click  . The Edit Scheduled Report dialog box appears.
3. Click the **Details** tab. The **Parameters** pane displays the parameters, which you can specify, that are available for the selected report;.
4. In the **Parameters** pane, provide the appropriate parameters. Options depend on the parameters available for the report. Possible options are:
   - Start Date field. Click  to open a calendar from which you select the start date of the report.
   - End Date field. Click  to open a calendar from which you select the end date of the report.
   - Job field. Type the name of the job for the report.
   - Header drop-down box. Select the header used for the job.
   - User drop-down box. Select the user running the job.
5. From the **Start Time** drop-down box, select the time on which the report is run.
6. In the **Recurrence Pattern** pane, specify when the report recurs.
   - Click the **Once** radio button to run the report once. After the report is run, rules for running the job are disabled.
   - Click the **Daily** radio button to specify that the report is run daily, and then specify the recurrence frequency.
   - Click the **Weekly** radio button to specify that the report is run weekly, and then specify the recurrence frequency.
   - Click the **Monthly** radio button to specify that the report is run monthly, and then specify the recurrence frequency.
7. Click the **Notifications** tab to configure notifications to be sent to recipients when certain reports actions occur. Refer to Configuring Notifications.
8. Click **OK**.

Deleting Scheduled Reports

To delete a scheduled report, do the following.

1. Select **Reports > Schedule**.
2. In the **Schedule Report** dialog box, select the scheduled report you want to delete and click  .

The scheduled report is deleted and is removed from the Schedule Report dialog box.

Viewing Reports

You can view reports in the View Reports dialog box.

Note You cannot view reports in tablet view.

If you open the dialog box from **Reports > View**, the search filters display the current date for all report rules and reports. If you open it from the Schedule Reports dialog box, the filters display the selected report rule.

These reports, in PDF format, are stored physically on the Perceptive DataTransfer server. You must specify the root directory of the location where the reports are stored. To do so, in the datatransfer.properties file, you must supply a *report_root* parameter. For example:

```
report_root=c:/reports
```

The PDF files are then stored in appropriate subdirectories within that root directory. To view a report, complete the following steps.

1. Perform one of the following actions.
   - Select **Reports > View**.
   - Select **Reports > Schedule** and click  .

The dialog box displays a table with the following information.

- Date: Date on which the report was created.
- Name: Name of the report.
- Report Rule Name: Name of the rule that caused the scheduled report to be run.
- Created By: Username of the administrator who configured the report.

You can filter the information in the View Reports dialog box by selecting the appropriate information in one or more of the following fields.

- Report Rule: Select the report rule for which to filter results.
- Reports: Select the report for which to filter results. Select *All* to specify all reports.
- Date Range fields: Click the  icon to open a calendar from which you can specify the start and end dates of a date range. You can choose both start and end dates or only a start or end date.

After specifying the filter information, click the **Filter** button to update the table with the selected criteria.

To view a report, click  . The report opens in a new tab or browser window in PDF format.

To permanently delete the report PDF file and the record of the report instance, click  . Reports are removed from the location specified in the *report_root* parameter in the datatransfer.properties file.

Configuring File Sources

You can configure file sources—servers or SFTP connections—for recurring jobs. The file source is the location that contains the input file that the recurring job uses. Recurring jobs run at a specified interval, using the input file from the server or SFTP site to which it can connect.

You can also configure file sources to download additional, non-input files from a specified location. For example, you can create a recurring job that automatically downloads the input file, creates a job for the file, downloads non-input files to be processed by the job, and places the non-input files in a location that Perceptive DataTransfer can access.

Refer to the following topics for more information.

- Adding File Sources
- Modifying File Sources
- Deleting File Sources
- Copying File Sources
- Testing File Sources
- Removing Sharing on Shared File Sources
- Rolling Back File Sources
- Obtaining the Latest Versions of File Sources
- Exporting File Sources
- Importing File Sources

Adding File Sources

You can add a new file source or add a file source that another workspace is sharing.

1. Select **Configuration > File Sources**.
2. Click  .
3. In the **Add File Source** dialog box, perform one of the following actions.
   - To add a new file source, complete the following steps.
     1. Click the **Add File Source** tab.
     2. In the **Name** field, type the name of the file source.
     3. From the **Type** drop-down box, select the type of file source (*Server* or *SFTP*).
   - To add a file source that another workspace is sharing, complete the following steps.
     1. Click the **Add Shared File Source** tab.
     2. From the **Workspace** drop-down box, select the workspace that is sharing the file source.
     3. From the **File Source** drop-down box, select the file source you want to add.
4. Click **OK**.

5. In the **Edit File Source** dialog box, perform one of the following actions.
   - To modify a server path file source, do the following.
     6. In the **Name** field, type the name of the file source.
     7. In the **Description** field, type descriptive text about the file source.
     8. In the **Path** field, type the path.
     9. In the **File Pattern Field**, type the file pattern information used to identify the input file located in the specified file source. Perceptive DataTransfer searches the file source for the input file based on the file pattern information specified.

     Use * to search for any number of characters; use ? to search for one character. For example:
     - **.xml* returns all xml files.
     - **.?ml* returns any files with an extension that has one character followed by *ml*.
     - *Application?.txt* returns all files that start with *Application*, that have a single character, and that have a *.txt* extension.

     Note Perceptive DataTransfer does not verify if the file pattern information is correct.
   - To modify an SFTP file source, complete the following steps.
     1. In the **Name** field, type the name of the file source.
     2. In the **Description** field, type descriptive text about the file source.
     3. In the **URL** field, type the URL of the SFTP connection.
     4. In the **Port** field, type the port number.
     5. In the **Path** field, type the SFTP connection path.
     6. In the **File Pattern** Field, type the file pattern information used to identify the input file located in the specified file source. Perceptive DataTransfer searches the file source for the input file based on the file pattern information specified.

     Use * to search for any number of characters; use ? to search for one character. For example:
     - **.xml* returns all xml files.
     - **.?ml* returns any files with an extension that has one character followed by *ml*.
     - *Application?.txt* returns all files that start with *Application*, that have a single character, and that have a *.txt* extension.
     7. In the **Username** field, type the username used to connect to the SFTP connection.
     8. In the **Password** field, type the password used to connect to the SFTP connection.

6. In the **File Downloads** pane, do the following.

    10. Check the **Additional Files** check box to specify that the file source has other, non-input files to download. When you create a recurring job that uses this file source, the input file is downloaded from the specified path or connection, and then additional files are downloaded from the same path or connection, given the parameters you specify in the File Downloads pane.
    11. In the **File Pattern** field, type the file pattern information used to identify the non-input file located in the file source.
    12. In the **Destination Path** field, type the location where the non-input files are placed when they are downloaded. The path must be on a Perceptive DataTransfer server or reachable from the Perceptive DataTransfer server.

7. In the **Source File Options** pane, specify how to manage the additional, non-input files in the file source location. Click one of the following options.
    - **Leave**. Performs no action on the files.
    - **Archive**. Archives the files and moves them to a location that you specify in the Archive Path field.
    - **Delete**. Permanently deletes the files from the file source.
8. Click **OK**.

The file source is added and appears in the File Source Administration dialog box.

Modifying File Sources

To modify a file source, do the following.

1. Select **Configuration > File Sources**.
2. In the **File Source Administration** dialog box, click the file source you want to modify and either click  or right-click the file source and select
3. In the **Edit File Source** dialog box, perform one of the following actions.
    - To modify a server path file source, complete the following steps.

        13. In the **Name** field, type the name of the file source.
        14. In the **Description** field, type descriptive text about the file source.
        15. In the **Path** field, type the path.
        16. In the **File Pattern Field**, type the file pattern information used to identify the input file located in the specified file source. Perceptive DataTransfer searches the file source for the input file based on the file pattern information specified.

        Use * to search for any number of characters; use ? to search for one character. For example:

        - **.xml* returns all xml files.
        - **.?ml* returns any files with an extension that has one character followed by *ml*.
        - *Application?.txt* returns all files that start with *Application*, that have a single character, and that have a *.txt* extension.

Note Perceptive DataTransfer does not verify if the file pattern information is correct.

- To modify an SFTP file source, complete the following steps.
  1. In the **Name** field, type the name of the file source.
  2. In the **Description** field, type descriptive text about the file source.
  3. In the **URL** field, type the URL of the SFTP connection.
  4. In the **Port** field, type the port number.
  5. In the **Path** field, type the SFTP connection path.
  6. In the **File Pattern Field**, type the file pattern information used to identify the input file located in the specified file source. Perceptive DataTransfer searches the file source for the input file based on the file pattern information specified.

  Use * to search for any number of characters; use ? to search for one character. For example:

  - **.xml* returns all xml files.
  - *.?ml* returns any files with an extension that has one character followed by *ml*.
  - *Application?.txt* returns all files that start with *Application*, that have a single character, and that have a *.txt* extension.

  7. In the **Username** field, type the username used to connect to the SFTP connection.
  8. In the **Password** field, type the password used to connect to the SFTP connection.

4. In the **File Downloads** pane, complete the following steps.
   17. Check the **Additional Files** check box to specify that the file source has other, non-input files to download. When you create a recurring job that uses this file source, the input file is downloaded from the specified path or connection, and then additional files are downloaded from the same path or connection, given the parameters you specify in the **File Downloads** pane.
   18. In the **File Pattern** field, type the file pattern information used to identify the non-input file located in the file source.
   19. In the Destination Path field, type the location where the non-input files are placed when they are downloaded. The path must be on a Perceptive DataTransfer server or reachable from the Perceptive DataTransfer server.
5. In the **Source File Options** pane, specify how to manage the additional, non-input files in the file source location. Click one of the following radio buttons.
   - **Leave**: Performs no action on the files.
   - **Archive**: Archives the files and moves them to a location that you specify in the Archive Path field.
   - **Delete**: Permanently deletes the files from the file source.
6. Click **OK**.

Deleting File Sources

To delete a file source, do the following.

1. Select **Configuration > File Sources**.
2. In the **File Source Administration** dialog box, select the file source you want to delete and either right-click it and select **Delete** or click . The file source is deleted and is removed from the **File Source Administration** dialog box.

Copying File Sources

To create a duplicate of a file source, complete the following steps.

1. Select Configuration > File Sources.
2. In the **File Source Administration** dialog box, select the file source you want to copy and either right-click it and select **Copy** or click . A copy of the file source is created and appears in the **File Source Administration** dialog box.

Testing File Sources

To test a file source connection, do the following.

1. Select **Configuration > File Sources**.
2. In the File Source Administration dialog box, select the file source you want to test and either right-click it and select **Test File Source** or click .
   - If the file source works properly, a dialog box appears and states that the test was successful.
   - If the file source does not work properly, a dialog box appears and indicates the reason that the test failed.

Removing Sharing on Shared File Sources

To remove sharing on a file source that is shared with another workspace, do the following.

1. Select **Configuration > File Sources**.
2. In the **File Source Administration** dialog box, select the file source for which you want to remove sharing and either right-click it and select **Remove Sharing** or click . Sharing is removed on the file source.

Rolling Back File Sources

If changes have been made and committed to a file source, you can use a previous version (*roll back*) of it.

1. Select **Configuration > File Sources**.
2. In the **File Source Administration** dialog box, click the file source you want to roll back and either right-click it and select **Rollback** or click . The **Rollback Manager** dialog box appears and displays the following information.
   - Name: Name of the file source.
   - Status: Status of the file source (*current* or *dated*).
   - Version Date: Date and time on which the version was created.
   - Created By: Username of the user who created the file source.
   - Comments: Comments that were added by the user.
3. Select the previous version to which you want to revert and click **Rollback**. The previous version is loaded.

Obtaining the Latest Versions of File Sources

If changes have been made to and committed to a file source, you can obtain the latest version of it.

1. Select **Configuration > File Sources**.
2. In the **File Source Administration** dialog box, click the file source for which you want to obtain the latest version and either right-click it and select
3. Get **Latest** or click .

Exporting File Sources

Note You cannot export file sources in tablet view.

You can export file sources to an XML file by doing the following.

1. Select **Configuration > File Sources**.
2. In the **File Source Administration** dialog box, select the file source you want to export.
3. Click and select **Export**.
4. Save the file to the appropriate location.

Importing Source Sources

Note You cannot import file sources in tablet view.

You can import file sources from an XML file by doing the following.

1. Select **Configuration > File Sources**.
2. In the **File Source Administration** dialog box, click and select **Import**.
3. In the **Import File Source** dialog box, click the **Browse** button and navigate to the file that you want to import or type the location of the file in the **File** field.

4. Check the **Update Existing Entities** check box to update an existing file source, with the same name as the file source you are importing, with the file source you are importing.

   If you do not check this check box, and there is an existing file source with the same name as the file source you are importing, *DUPLICATE-* is appended to the start of the name of the file source you are importing; the existing file source is not updated. If a file source exists with the name DUPLICATE-

   <file_source_name>, *DUPLICATE(2)-* is appended to the start of the name. The number continues to be incremented so that all file sources have unique names.

5. Click **OK**.

Configuring data providers

You can configure Perceptive DataTransfer to use web service or SOAP web service data providers for communication.

Note: SOAP web service data providers are supported in Perceptive DataTransfer 6.4.5 and higher.

- Configuring web service data providers
- Configuring SOAP web service data providers
- Testing data providers
- Configuring data provider program functions
- Deleting data providers.
- Copying data providers
- Removing sharing on data providers
- Rolling back data providers.
- Obtaining the latest versions of data providers
- Exporting data providers
- Importing data providers

Configuring web service data providers

You can configure Perceptive DataTransfer to use web service data providers for communication. Perform the following steps to configure a web service data provider.

1. Add and configure a web service connection.
2. To provide the appropriate permissions to configure the web service data provider, enable the **Edit Data Providers** privilege in **Administration > User**.
3. Add a web service data provider.
4. Configure the web service data provider.
5. Test the web service data provider.
6. Create a web service header.
7. Create a job that uses the web service connection and header.
8. Configure the code to run the web services.

Adding web service data providers

To add a web service data provider, complete the following steps.

1. Select **Configuration > Data Providers.**
2. In the **Data Provider Administration** box, click  .
3. In the **Add Data Provider**, dialog box, complete one of the following actions.
   - To add a new data provider, complete the following substeps.
     1. Click the **Add Data Provider** tab.
     2. In the **Name** field, type the name of the data provider.
     3. Select **Web Service** from the **Type** drop-down box.
   - To add a data provider that another workspace is sharing, complete the following substeps.
     1. Click the **Add Shared Data Provider** tab.
     2. From the **Workspace** drop-down box, select the workspace that is sharing the data provider.
     3. Select the data provider from the **Data Provider** drop-down box.
     4. Click **OK**.

Modifying web service data providers

When you save a web service data provider, only the connection name is saved; the URL associated with the connection is not saved so that you can configure only one web service data provider and use the appropriate connection, depending on the connection group chosen, for two different servers.

For example, you may have the following servers and groups.

- Both test and production servers that use the web service.
- A test connection group with a web service connection using the test URL.
- A production connection group with a web service using the production URL.

If you name both connections *DTSIS* and use the DTSIS connection, you can use the same data provider for both URLs.

For example, you save a data provider with DTSIS as the connection. The next time you open the data provider to edit it, the application checks the connection groups in alphabetical order and searches for a connection with the name *DTSIS*. If a connection with that name is found, that connection and its associated connection group automatically display in the corresponding drop-down boxes. If the connection is not found, the Connection Group and Connection drop-down boxes do not automatically display any data and the message `A valid web service connection must be selected` appears in the **Connection URL** field.

To modify a web service data provider, complete the following steps.

1. Select **Configuration > Data Providers**.
2. In the **Data Provider Administration** dialog box, select the web service data provider you want to modify and click  .
3. In the **Edit Web Service Data Provider** dialog box, in the **Name** field, type the name of the web service data provider.

4. In the **Description** field, type descriptive text about the data provider.
5. From the **Connection Group** drop-down box, select the connection group that contains the connection you want to use for the web service data provider. The connection groups that you have permission to view and use display in the drop-down box.
6. From the **Connection** drop-down box, select the connection, which belongs to the connection group you selected, to use for the data provider.

   After you select a connection, the **Connection URL** field in the right-top pane displays the URL for the connection. You cannot modify this field. If you do not provide a valid connection, the field displays the text **A valid web service connection must be selected.**

1. From the **HTTP Method** field, select the HTTP method used by the web service request (*GET*, *POST*, *PUT*, or *DELETE*).
2. Click the **URL Suffix** tab and type the URL suffix to the web service being requested. The URL suffix is the portion of the web service request that is dynamic (the parameters). When the web service runs, the connection URL and the URL suffix are combined to obtain the full web service URL.
3. Click the **Payload** tab to add a request body, if necessary.
   - If you are using JSON, type either the URL query string to use after you connect to the web service or a JSON-formatted object.
   - If you are using XML, type either the URL query string to use after you connect to the web service or an XML-formatted request.
4. In the **REST Type** field, click the radio button for the REST web service type used by the web service request (XML or JSON).
5. To map Perceptive DataTransfer program variables to web service input and output parameters, drag and drop variables from the **Program Variables** pane to the **Input** and **Output** tabs.

   When you add an input or output parameter, the name of the program variable is the name of the input or output parameter and it appears in the **Name** column. For example, if you map the `$ws_id` program variable to an input parameter, the input parameter is named `$ws_id`, which appears in the **Name** column.
6. To modify an input value, click the **Input** tab, double-click in the field, and type the appropriate information.
   - **Description**: Type descriptive text about the parameter.
   - **Location**: Ordinal value of the field. The first field has a value of {0} (zero), the second field has a value of {1}, and so on
   - **Substitution Expression**: Type the values for the input fields. You must always use {0} for the substitution value. When you run the web service, the value of the program variable is substituted for {0}.
7. To modify an output value, click the **Output** tab, double-click the field, and type the appropriate information.
   - **Description**: Type descriptive text about the parameter.
   - **Query**: Type the XPath (if you are using XML) or JSONPath (if you are using JSON) query used to retrieve information from the results of the web service request.
8. Click **OK**.

Configuring SOAP web service data providers

You can configure Perceptive DataTransfer to use SOAP web service data providers for communication. Perform the following steps to configure a SOAP web service data provider.

1. Add and configure a SOAP web service connection. Refer to Adding and modify SOAP web service connections.
2. To provide the appropriate permissions to configure the SOAP web service data provider, enable the **Edit Data Providers** privilege in **Administration > User**. For more information about privileges, refer to Configuring Users and Privileges.
3. Add a SOAP web service data provider. Refer to Adding and modify SOAP web service connections.
4. Configure the SOAP web service data provider. Refer to Modifying SOAP web service data providers.
5. Test the SOAP web service data provider. Refer to Testing data providers.
6. Create a SOAP web service header. Refer to Configuring and modifying web service headers.
7. Create a job that uses the SOAP web service connection and header. Refer to Configuring Jobs and Processing Records.
8. Configure the code to run the SOAP web services. Refer to Configuring data provider program functions.

Adding SOAP web service data providers

To add a SOAP web service data provider, complete the following steps.

1. Select **Configuration > Data Providers**.
2. In the **Data Provider Administration** dialog box, click  .
3. In the **Add Data Provider** dialog box, complete one of the following actions.
   - To add a new data provider, complete the following substeps.
     1. Click the **Add Data Provider** tab.
     2. In the **Name** field, type the name of the data provider.
     3. Select **SOAP Web Service** from the **Type** drop-down box.
   - To add a data provider that another workspace is sharing, complete the following substeps.
     1. Click the **Add Shared Data Provider** tab.
     2. From the **Workspace** drop-down box, select the workspace that is sharing the data provider.
     3. Select the data provider from the **Data Provider** drop-down box.
4. Click **OK**.

Modifying SOAP web service data providers

To modify a SOAP web service data provider, complete the following steps.

1. Select **Configuration > Data Providers**.

2. In the **Data Provider Administration** dialog box, select the SOAP web service data provider you want to modify and click  .
3. In the **Edit SOAP Web Service Data Provider** dialog box, in the **Name** field, type the name of the SOAP web service data provider.
4. In the **Description** field, type descriptive text about the data provider.
5. From the **Connection Group** drop-down box, select the connection group that contains the connection you want to use for the SOAP web service data provider. The connection groups that you have permission to view and use display in the drop-down box.
6. From the **Connection** drop-down box, select the connection, which belongs to the connection group you selected, to use for the data provider.

   After you select a connection, the **URL Endpoint** field in the right-top pane displays the URL endpoint for the connection. You cannot modify this field. If you do not provide a valid connection, the field displays the text **A valid web service connection must be selected.**

1. In the **SOAP Action** field, specify the SOAP web service method that this configuration requests.
2. From the **HTTP Method** drop-down box, select the HTTP method used by the SOAP web service request (*GET*, *POST*, *PUT*, or *DELETE*).
3. In the **SOAP Message** field, enter the XML text of the SOAP message. You can optionally use the {#} notation to denote portions of XML that should be substituted at runtime with the values of the corresponding program variables.

   Example

   ```
   <m13:COMMON_ID>{2}</m13:COMMON_ID>

   <m13:SEQ_3C>{3}</m13:SEQ_3C>
   ```

   The {2}, and {3} are substituted at runtime. You can map program variables to the occurrences of the # notation, substituting the value of the program variable into the occurrence. For example, if you map the *$ps1* variable to {2}, at runtime, the value of the $ps1 variable is inserted in the XML file between the <m13:COMMON_ID> and </m13:COMMON_ID> tags. To map program variables, refer to the following step.

4. To map Perceptive DataTransfer program variables to web service input and output parameters, drag and drop variables from the **Program Variables** pane to the **Input** and **Output** tabs.

   When you add an input or output parameter, the name of the program variable is the name of the input or output parameter and it appears in the **Name** column.

5. To modify an input value, click the **Input** tab, double-click the field, and type the appropriate information.

   - **Description**: Type descriptive text about the parameter.
   - **Location**: Ordinal value of the field. The first field has a value of {0} (zero), the second field has a value of {1}, and so on
   - **Substitution Expression**: Type the values for the input fields. You must always use {0} for the substitution value. When you run the web service, the value of the program variable is substituted for {0}.

6. To modify an output value, click the Output tab, double-click the field, and type the appropriate information.
   - **Description**: Type descriptive text about the parameter.
   - **Query**: Type the XPath or JSONPath query you use to retrieve information from the results of the SOAP web service request.
7. Click **OK**.

Testing data providers

To test a data provider, complete the following steps.

1. Select **Configuration > Data Providers**.
2. In the **Data Provider Administration** dialog box, select the data provider you want to test and click  .
3. In the **Edit Web Service Data Provider** or **Edit SOAP Web Service Data Provider** dialog box, click **Test**.
4. In the **Web Service Test** dialog box, in the **Input Data** pane, double-click the **Value** column and type a value to test.
5. Click **Execute**. The following occurs.
   - The **Request URL** tab displays the full URL used for the web service. If you did not select a web service connection when testing the web service, the test does not work and the **Request URL** tab displays a `Malformed URL` message.
   - The **Output** field displays the values of the output parameters.
   - The **Request Payload** tab displays the full payload sent in the request.
   - The **Raw Response** pane displays the HTTP response code and the raw XML or JSON returned in the response.

Configuring data provider program functions

After you configure a web service or SOAP web service connection and a web service or SOAP web service data provider, you can run these web services in the code using the following *DataProviderCall*, *DataProviderPrompt,* and *begin-dataprovider* keywords.

Understanding the DataProviderCall function

This function uses two parameters.

- The `$webServiceName` parameter is the name of the web service to run.
- The optional `$payload` parameter replaces the existing payload value of the web service before it runs. Using this parameter, you can dynamically create the payload and pass it to the web service at runtime.

Use the following syntax.

```
DataProviderCall ($webServiceName, $payload)
```

For example:

```
let $ws_id = '10'

DataProviderCall('personWS')
```

In this example, personWS is the name of the web service you run. When Perceptive DataTransfer reaches this line of code, it performs the following tasks to run the personWS web service.

1. Perceptive DataTransfer looks up the web service by the supplied data provider name.
2. It verifies the current connection group (the group configured for the job configured to run) for the saved connection name.
3. Perceptive DataTransfer finds the connection for the connection group. One of the following situations occurs.
   - If it finds the matching connection, it uses the URL to run the web service request; then, it populates the input parameters with the values of the mapped program variables. If the web service returns more than one result, the record suspends with a *Too many results returned* reason.
   - If it does not find a matching connection, the web service request results in an error.

In the preceding example, after you run the personWS web service, the output values are inserted into the mapped program variables. The value of the `$ws_first_name` parameter is `John`, the value of the `$ws_last_name` parameter is `Doe`, and the value of the `$ws_id` parameter is `10`.

For detailed information about programming concepts and syntax, refer to Appendix A: Programming Concepts.

Understanding the DataProviderPrompt function

The command runs the specified web service, as described in Understanding the DataProvider Call function. It also displays a prompt that users can use to view results.

Use the following syntax.

```
DataProviderPrompt $prompt, $webServiceName, $hidePromptWhenEmpty, $autoSelectResult,
$payload
```

Example

```
let $ws_id = ' '

let $ws_first_name = ' '

let $ws_last_name = ' '

DataProviderPrompt 'Choose the matching person.', 'personWS'
```

This function uses two required parameters: a prompt (`$prompt`) and the name of the web service (`$webServiceName`).

You can also set two optional string parameters (for example, string literals, string expressions, string variables, and so on). Use a `Y` (or `y`) value to indicate that the option is enabled. You can also use the optional `$payload` parameter. The optional parameters are listed below.

- The `$hidePromptWhenEmpty` parameter is a flag to hide the prompt if no results are returned by the data provider—if the request had no results, Perceptive DataTransfer does not prompt the user to select one.

Example

```
let $ws_id = ' '
let $ws_first_name = ' '
let $ws_last_name = ' '
DataProviderPrompt 'Choose the matching person.', 'personWS', 'Y'
```

- The `$autoSelectResult` parameter is a flag to automatically select the result if exactly one result is returned by the data provider—if the data provider returns a single record, the record is auto-selected and no prompt displays.

Example

```
let $ws_id = ' '
let $ws_first_name = ' '
let $ws_last_name = ' '
DataProviderPrompt 'Choose the matching person.', 'personWS', 'Y', 'Y'
```

- The optional `$payload` parameter replaces the existing payload value of the web service before it runs. Using this parameter, you can dynamically create the payload and pass it to the web service at runtime.

In the **Result Selection** dialog box, the prompt you supply appears in the field at the top of the dialog box. All results returned by the web service appear in the table below.

After a user selects a row, the **OK** button is enabled. If the user clicks **OK**, all the values from the selected row are placed into the mapped program variables.

This prompt appears when a job is running in interactive mode only. If this function is processed when a job is run in batch mode, Perceptive DataTransfer checks the number of return results. The web service performs the following actions.

- If there are no results, the output program variables are set to the empty string.
- If there is one result, the output values are placed into the mapped program variables.
- If there is more than one result, the record is suspended with a `Too may results` reason.

Understanding the begin-dataprovider function

You can use a begin-dataprovider/end-dataprovider block of code to run a specified data provider and iterate over its results. Use the following syntax.

```
begin-dataprovider $webServiceName
```

For example, consider the following begin-select syntax.

```
begin-select
       nu_id  ~#pers_nuid let
       #nu_id = #pers_nuid let
       $status = 'M'
       let $mtch_code = 'MP1'
       store 1, '#nu_id'   ! rank, ID - will be fed to review process
       stop   ! exit match process here
       FROM nu_person  WHERE ssn = ~$f_ssn
   end-select
```

The preceding example could use a data provider instead of a SQL query, as shown in the following example.

```
begin-dataprovider 'personWS'
       let $status = 'M'
       let $mtch_code = 'MP1'
       store 1, '$ws_id'   ! rank, ID - will be fed to review process
       stop   ! exit match process here
   end-dataprovider
```

After `begin-dataprovider`, you must provide the name of the data provider to run. In the preceding example, `personWS` is the name of the data provider.

The same code can be used inside `begin-select` as it is inside `begin-dataprovider`. Also, similar to `begin-select`, `begin-dataprovider` runs the data provider and iterates over the results, running the lines of codes for each result.

Like an exit-select statement that exits the begin-select/end-select code block, `exit-dataprovider` exits the begin-dataprovider/end-dataprovider code.

You can use the optional `$payload` parameter to replace the existing payload value of the web service before it runs. Using this parameter, you can dynamically create the payload and pass it to the web service at runtime. Use the following syntax.

```
begin-dataprovider $webServiceName $payload
```

Deleting data providers

To delete a data provider, complete the following steps.

1. Select **Configuration > Data Providers**.
2. In the **Data Provider Administration** dialog box, select the data provider you want to delete and click  .

The data provider is deleted and is removed from the **Data Provider Administration** dialog box.

Copying data providers

To create a duplicate of a data provider, complete the following steps.

1. Select **Configuration > Data Providers**.
2. In the **Data Provider Administration** dialog box, select the data provider you want to copy and click  .

A copy of the data provider is created and appears in the **Data Provider Administration** dialog box.

Removing sharing on data providers

To remove sharing on a data provider that is shared with another workspace, complete the following steps.

1. Select **Configuration > Data Providers**.
2. In the Data Provider Administration dialog box, select the data provider for which you want to remove sharing and click  .

Sharing is removed on the data provider.

Rolling back data providers

If changes are made and committed to a data provider, complete the following steps to obtain a previous version.

1. Select **Configuration > Data Providers**.
2. In the **Data Provider Administration** dialog box**,** select the data provider you want to roll back and click  .
3. The **Rollback Manager** dialog box appears and displays the following information.
   - **Name**: Name of the data provider.
   - **Status**: Status of the data provider (*current* or *dated*).
   - **Version Date**: Date and time on which the version was created.
   - **Created By**: User name of the user who created the data provider.
   - **Comments**: Comments that were added by the user.
4. Select the previous version to which you want to revert and click **Rollback**.

The previous version is loaded.

Obtaining the latest versions of data providers

If changes have been made to and committed to a data provider, you can obtain the latest version of it by completing the following steps.

1. Select **Configuration > Data Providers**.
2. In the **Data Provider Administration** dialog box, select the data provider for which you want to obtain the latest version and either right-click it and select **Get Latest** or click  .

Exporting data providers

To export a data provider to an XML file, complete the following steps.

Note You cannot export data providers in tablet view.

1. Select **Configuration > Data Providers**.
2. In the **Data Provider Administration** dialog box, select the data provider you want to export.
3. Click  and select **Export**.
4. Save the file to the appropriate location.

Importing data providers

To import a data provider from an XML file, complete the following steps.

Note You cannot import data providers in tablet view.

1. Select **Configuration > Data Providers**.
2. In the **Data Provider Administration** dialog box, click  and select **Import**.
3. In the **Import Data Provider** dialog box, click the **Browse** button and navigate to the file that you want to import or type the location of the file in the **File** field.
4. Select the **Update Existing Entities** check box to update an existing data provider with the same name as the data provider you are importing.

   If you do not select this check box and there is an existing data provider with the same name as the data provider you are importing, **DUPLICATE-** is appended to the start of the name of the data provider you are importing; the existing data provider is not updated. If a data provider exists with the name **DUPLICATE-<*data_provider_name*>**, **DUPLICATE(2)**- is appended to the start of the name. The number continues to be incremented so that all data providers have unique names.
5. Click **OK**.

Configuring Document Templates

You can configure document templates that can be used in the NQL code to create files based on the template definition for uploading to Nolij Web or ImageNow.

Adding and Modifying Document Templates

To add a document template, do the following.

1. Select **Configuration > Document Templates**.
2. In the **Document Administration** dialog box, perform one of the following actions.
   - To add a new document template, click  to display the **Add Document Template** dialog box and perform one of the following actions.
     - To add a new document template, click the **Add Document Template** tab and type a name for the template in the Name field.
     - To add a document template that another workspace is sharing, click the **Add Shared Document Template** tab. Then, from the **Workspace** drop-down box, select the workspace that is sharing the document template, and select the template from the **Document Template** drop-down box.
   - To modify a document template, select the template you want to modify and either right-click it and select **Edit** or click  to display the **Edit Document Template** dialog box.
3. Click **OK**.

   Note In the name field, type the name of the document template.
4. In the **Description** field, type descriptive text about the template.
5. In the **Type** field, type the file type (the file extension) for the file created from the template.
6. In the **Template** field, click **Browse** and navigate to the location of the document template you want to upload.

   Note You cannot upload document templates in tablet view.

   The contents of the file appears in the text field below the Browse... button, and the Type field displays the type of the uploaded document template. You can modify both fields.

   Document template text can contain program variables. When a document is created from the template, the variables are replaced with their values, creating a dynamic, context-sensitive document. Use the PopulateTemplate function in the NQL to create such a document (for more information, refer to Understanding the PopulateTemplate faction).
7. Click **OK**.

Deleting Document Templates

To delete a document template, do the following.

1. Select **Configuration > Document Templates.**
2. In the **Document Template Administration** dialog box, select the document template you want to delete and either right-click it and select **Delete** or click .

The document template is deleted and is removed from the Document Template Administration dialog box.

Copying Document Templates

To create a duplicate of a document template, do the following.

1. Select **Configuration > Document Templates.**
2. In the **Document Template Administration** dialog box, select the document template you want to copy and either right-click it and select **Copy** or click .

A copy of the document template is created and appears in the Document Template Administration dialog box.

Removing Sharing on Document Templates

To remove sharing on a document template that is shared with another workspace, do the following.

1. Select **Configuration > Document Templates.**
2. In the **Document Template Administration** dialog box, select the document template for which you want to remove sharing and either right-click it and select **Remove Sharing**, or click .

Sharing is removed on the document template.

Rolling Back Document Templates

If changes have been made and committed to a document template, you can use a previous version (*roll back*) of it.

1. Select **Configuration > Document Templates.**
2. In the **Document Template Administration** dialog box, click the document template you want to roll back and either right-click it and select **Rollback** or click .
3. The **Rollback Manager** dialog box appears and displays the following information.
   - Name: Name of the document template.
   - Status: Status of the document template (*current* or *dated*).
   - Version Date: Date and time on which the version was created.
   - Created By: Username of the user who created the document template.
   - Comments: Comments that were added by the user.
4. Select the previous version to which you want to revert and click **Rollback**. The previous version is loaded.

Obtaining the Latest Versions of Document Templates

If changes have been made to and committed to a document template, you can obtain the latest version of it.

1. Select **Configuration > Document Templates.**
2. In the **Document Template Administration** dialog box, click the document template for which you want to obtain the latest version and either right-click it and select **Get Latest** or click .

Exporting Document Templates

Note You cannot export document templates in tablet view.

You can export a document template to an XML file by doing the following.

1. Select **Configuration > Document Templates.**
2. In the **Document Template Administration** dialog box, select the document template you want to export.
3. Click and select **Export**.
4. Save the file to the appropriate location.

Importing Document Templates

Note You cannot import document templates in tablet view.

You can import a document template from an XML file by doing the following.

1. Select **Configuration > Document Templates.**
2. In the **Document Template Administration** dialog box, click and select **Import**.
3. In the Import Document Template dialog box, click the **Browse** button and navigate to the file that you want to import or type the location of the file in the **File** field.
4. Check the **Update Existing Entities** check box to update an existing document template, with the same name as the document template you are importing, with the document template you are importing.

   If you do not check this check box, and there is an existing document template with the same name as the document template you are importing, *DUPLICATE-* is appended to the start of the name of the document template you are importing; the existing document template is not updated. If a document template exists with the name DUPLICATE-<*document_template_name*>, *DUPLICATE(2)-* is appended to the start of the name. The number continues to be incremented so that all document templates have unique names.
5. Click **OK**.

Configuring ImageNow Index Maps

You can define ImageNow index maps, which are used when uploading ImageNow documents to ImageNow, using Perceptive DataTransfer. Index maps obtain document information from the specified integration server connection and link ImageNow document properties to program variables in the code. Use the ImageNowAddDoc syntax in the code to configure parameters used by Perceptive DataTransfer to upload ImageNow documents.

For more information about integration server connections, refer to Adding and modifying integration server connections. For more information about the ImageNowAddDoc syntax, refer to Understanding the ImageNowAddDoc function.

Refer to the following topics for more information.

- Adding and Modifying ImageNow Index Maps
- Deleting ImageNow Index Maps
- Copying ImageNow Index Maps
- Removing Sharing on ImageNow Index Maps
- Rolling Back ImageNow Index Maps
- Obtaining the Latest Versions of ImageNow Index Maps
- Exporting ImageNow Index Maps
- Importing ImageNow Index Maps

Adding and Modifying ImageNow Index Maps

Note You cannot add ImageNow index maps in tablet view. To add an ImageNow index map, do the following.

1. Select **Configuration > ImageNow**.
2. In the **ImageNow Index Map Administration** dialog box, perform one of the following actions.
   - To add a new index map, click . This button does not appear in tablet view; you cannot add ImageNow index maps in tablet view.
     1. In the **Add ImageNow** dialog box, perform one of the following actions.
        - To add a new index map, click the **Add ImageNow** tab. Then, in the **Name** field, type the name of the index map.
        - To add an index map that another workspace is sharing, click the **Add Shared ImageNow** tab. Then, from the **Workspace** drop-down box, select the workspace that is sharing the index map, and select the index map from the **ImageNow Entity** drop-down box.
     2. Click **OK**.
   - To modify an index map, select the index map and either right-click it and select **Edit** or click . The **Edit ImageNow Index** Map dialog box appears.

3. From the **Connection Group** drop-down box, select the connection group that contains the integration server connection to use. This connection group is not saved with an association to the index map; it is used to obtain custom ImageNow document properties as program variables for the index map.

   For more information about adding integration server connections, refer to Adding and modifying integration server connections.

4. In the **Name** field, type the name of the index map. The name must be unique.
5. In the **Description** field, type descriptive text about the index map.
6. In the **Drawer** field, specify the ImageNow drawer that contains the ImageNow documents by dragging and dropping the appropriate program variable from the **Program Variables** pane. This field is required.

   To remove an entry, right-click the field and select **Clear**.

7. In the **Document Type** field, specify the ImageNow document type, contained in the ImageNow drawer, to which to link to the ImageNow index map by dragging and dropping the appropriate program variable from the **Program Variables** pane. This field is required.

   To remove an entry, right-click the field and select **Clear**.

8. In the **Field 1** through **Field 5** fields, define the index fields for the document by dragging and dropping the appropriate program variables from the **Program Variables** pane to each field. You can also leave each field empty.

   When the **ImageNowAddDoc** function executes, the code inserts these fields with the values contained in the specified program variables.

9. In the **Custom Properties** pane, add or modify the ImageNow custom properties associated with the index map by completing the following substeps.

   1. To add a new property, click  .

   The **Select Custom Properties** dialog box appears. It displays all available ImageNow custom properties, excluding any that you have already added to the index map.

   2. Select one or more properties to add to the index map. To select multiple properties:

   - Click a property and press and hold the **Ctrl** key as you click each property with your mouse.
   - Click a property and press and hold the **Shift** key as you click another property to select contiguous properties.

   3. Click **OK**. The selected properties appear in the **Custom Properties** pane.
   4. Link program variables to the custom properties by selecting a program variable in the **Program Variables** pane and dragging and dropping it to the appropriate custom property.

10. To delete a custom property from the ImageNow index map, select the property you want to delete in the Custom Properties pane and click the **Delete** button or right-click it and select **Delete**.

Use the ImageNowAddDoc function in your code to configure the parameters used by Perceptive DataTransfer to upload ImageNow documents. For more information, refer to Understanding the ImageNowAddDoc function.

Deleting ImageNow Index Maps

To delete an index map, complete the following steps.

1. Select **Configuration > ImageNow**.
2. In the **ImageNow Index Map Administration** dialog box, select the index map you want to delete and either right-click it and select **Delete** or click .

The index map is deleted and is removed from the ImageNow Index Map Administration dialog box.

Copying ImageNow Index Maps

To create a duplicate of an ImageNow index map, do the following.

1. Select **Configuration > ImageNow**.
2. In the **ImageNow Index Map Administration** dialog box, select the index map you want to copy and either right-click it and select **Copy** or click .

A copy of the index map is created and appears in the ImageNow Index Map Administration dialog box.

Removing Sharing on ImageNow Index Maps

To remove sharing on an index map that is shared with another workspace, complete the following steps.

1. Select **Configuration > ImageNow**.
2. In the ImageNow Index Map Administration dialog box, select the index map for which you want to remove sharing and either right-click it and select Remove Sharing, or click .

Sharing is removed on the index map.

Rolling Back ImageNow Index Maps

If changes have been made and committed to an ImageNow index map, you can use a previous version (*roll back*) of it.

1. Select **Configuration > ImageNow**.
2. In the **ImageNow Index Map Administration** dialog box, click the index map you want to roll back and either right-click it and select **Rollback** or click . The **Rollback Manager** dialog box appears and displays the following information.
   - Name: Name of the ImageNow index map.
   - Status: Status of the index map (*current* or *dated*).
   - Version Date: Date and time on which the version was created.
   - Created By: Username of the user who created the index map.
   - Comments: Comments that were added by the user.
3. Select the previous version to which you want to revert and click **Rollback**.

The previous version is loaded.

Obtaining the Latest Versions of ImageNow Index Maps

If changes have been made to and committed to an index map, you can obtain the latest version of it.

1. Select **Configuration > ImageNow**.
2. In the **ImageNow Index Map Administration** dialog box, click the index map for which you want to obtain the latest version and either right-click it and select **Get Latest** or click .

Exporting ImageNow Index Maps

Note You cannot export ImageNow index maps in tablet view.

You can export an ImageNow index map to an XML file by doing the following.

1. Select **Configuration > ImageNow**.
2. In the **ImageNow Index Map Administration** dialog box, select the index map you want to export.
3. Click and select **Export**.
4. Save the file to the appropriate location.

Importing ImageNow Index Maps

Note You cannot import ImageNow index maps in tablet view.

1. Select **Configuration > ImageNow**.
2. In the **ImageNow Index Map Administration** dialog box click and select **Import**.
3. In the **Import ImageNow Index Map** dialog box, click the **Browse** button and navigate to the file you want to import or type the location of the file in the **File** field.
4. Check the Update Existing Entities check box to update an existing ImageNow index map, with the same name as the ImageNow index map you are importing, with the ImageNow index map you are importing.

   If you do not check this check box, and there is an existing ImageNow index map with the same name as the ImageNow index map you are importing, DUPLICATE- is appended to the start of the name of the ImageNow index map you are importing; the existing ImageNow index map is not updated. If an ImageNow index map exists with the name DUPLICATE-<index_map_name>, DUPLICATE(2)- is appended to the start of the name. The number continues to be incremented so that all ImageNow index maps have unique names.
5. Click **OK**.

Using Perceptive DataTransfer Tools

Refer to the following topics for more information.

- Configuring Lookup Values
- Configuring Names to Check
- Viewing Log Messages
- Configuring Data Exports
- Viewing the Import History

Configuring Lookup Values

Use the lookup tool to specify the data you want to map from the input file to the database.

You configure the Perceptive DataTransfer logic to substitute the output value for the input value and store the output value in your database.

For example, an input file may have a value of *Humanities* for a department; its corresponding database (output) value is *HUM*. You can use the lookup tool to map *Humanities* as *HUM* and then configure code to store *Humanities* as *HUM* in your database.

Example syntax for configuring the code is:

```
let $some_var = lookup("MAJOR", $MAJR)
```

When a job is run, lookup mapping for *MAJOR* determines that the correct output value for *Humanities* is *HUM*. The variable *$some_var* contains the value *HUM*. *$MAJR* contains *Humanities*.

For more information about configuring procedures, refer to Configuring Procedures. For detailed information about configuring logic, refer to Appendix A: Programming Concepts.

You can export lookups to an XML file, and you can import lookups from either a Perceptive DataTransfer XML file or a delimited file.

To open the Lookup Administration dialog box, either select **Tools > Lookup** or click the **Lookup** button in the window that appears when you open or edit jobs. When you click the Lookup button, the Lookup Administration dialog box displays the lookup entries for the header that is being used for the job.

You can filter the entries in the Lookup Administration dialog box by header, by field, or by both. Select the appropriate header from the Header drop-down box to view entries for that header. Select the appropriate field from the Field drop-down box to view entries for that field.

Adding Lookup Entries

To add a lookup entry, do the following.

1. Select **Tools > Lookup**.
2. In the **Lookup Administration** dialog box, click  .
3. In the **Add Lookup** dialog box, from the **Header** drop-down box, optionally select the header for which to configure the lookup entry. Select **No Header** if you want to use this entry for all headers.

4. From the **Field** drop-down box, select the header input variable to use or type the input variable in the **Field** field. This variable corresponds to the field in the input file that you want to map to a database valid code (output).
5. In the **In** field, type the value of the input variable to map to the output value in the **Out** field.
6. In the **Out** field, type the output value to map to the input value (specified in the **In** field).
7. Click **Apply**.
8. Click **OK**.

Modifying Lookup Entries

To modify a lookup entry, do the following.

1. Select **Tools > Lookup**.
2. In the Lookup Administration dialog box, select a lookup entry to modify and either right-click it and select **Edit** or click .
3. In the **Edit Lookup** dialog box, in the **In** field, type the input value, which is the value of the input variable to map to the database valid code in the **Out** field.
4. In the **Out** field, type the database valid code to map to input value (specified in the **In** field).
5. Click **OK**.

Deleting Lookup Entries

To delete a lookup entry, do the following.

1. Select **Tools > Lookup**.
2. In the Lookup Administration dialog box, select a lookup entry to modify and either right-click it and select **Delete** or click .

The entry is deleted and is removed from the Lookup Administration dialog box.

Importing Lookup Files

Note You cannot import lookup files in tablet view.

To import lookup entries from a either a Perceptive DataTransfer file or a delimited file, do the following.

1. Select **Tools > Lookup**.
2. In the **Lookup Administration** dialog box, click .
3. In the **Import Lookups** dialog box, perform one of the following actions.
   - To import a delimited file, do the following.
     1. Click the **Delimited** tab.
     2. From the **Header** drop-down box, select the header file to use.

3. From the **Field** drop-down box, select the header input field or type a new field to create it. This variable corresponds to the field in the input file that you want to map to a database valid code (output).
4. Specify how the imported file is delimited; either click the **Tab** radio button or click the **Specify** option and then provide a value in the Specify field.
5. In the **Upload File** field, click **Browse** and navigate to the file you want to import.
6. Click **Upload**.

4. To import a Perceptive DataTransfer file, click the **Transfer File** tab. Then, in the **File** field, click **Browse**, navigate to the file you want to import, and select it.
5. Click **OK**.

The file is imported, and imported lookup values are displayed in the Lookup Administration dialog box.

Exporting Lookup Files

Note You cannot export lookup files in tablet view. To export lookups as an XML file, do the following.

1. Select **Tools > Lookup**.
2. In the Lookup Administration dialog box, click  .
3. Save the file in the appropriate location.

Configuring Names to Check

You can use the Check Name tool to configure different output values that an input name may have. You can substitute this output value for the input name within the Perceptive DataTransfer logic.

For example, for an input name *Alexandria*, you could use *Alex* or *Lexie* as substitutions. Code syntax is:

```
CHECKNAME inCNamevar, inFNamevar, outCntvar, outNamevar1, outNamevar2,... outNamevarN
```

- The first two variables (*inCNamevar* and *inFNamevar*) are input variables for which you are substituting values. inFNamevar is an optional parameter.
- The third variable (*outCntvar*) specifies the number (count) of output variables.
- Variables that appear after the third variable (*outCntvar)* are the output variables.

When the Perceptive DataTransfer logic is run, it searches for matches between the input and output variables. If values for output variables are found, the output variables are populated with all variations of the input name. If no matches are found, the input name is assigned to the output variables.

For more information about configuring procedures, refer to Configuring Procedures. For detailed information about logic, refer to Appendix A: Programming Concepts.

You can import check name values from and export values to an XML file.

To open the Check Name Administration dialog box, select **Tools > Check Names**.

Refer to the following topics for more information

- Adding and Modifying Names
- Deleting Names
- Importing Names

- Exporting Names

Adding and Modifying Names

To add or modify a name, complete the following steps.

1. Select **Tools > Check Names**.
2. In the **Check Name Administration** dialog, perform one of the following actions.
   - To add a new name, click  . The **Add New Check Name** dialog box appears.
   - To modify a name, select the name and click  . The **Edit Check Name** dialog box appears.
3. To add a new value to the name, complete the following substeps.
   1. Click  to display the **New Check Name Value** dialog box appears.
   2. In the **Value** field, type the value that the name can have.

   Note You cannot add a value that is already configured for another entry. You can only use a value once.
   3. Click **Apply** to add the value to the check name entry.
   4. Continue adding values for the name, and then click **OK**.
8. To modify a value, complete the following steps.
   5. Select the value you want to modify and either right-click it and select **Edit** or click  .
   6. In the **Edit** dialog box, type the value in the appropriate field and click **OK**.
9. To delete a value, select the value you want to delete and either right-click it and select **Delete** or click  .

The value is deleted and is removed from the dialog box.

Deleting Names

To delete a name, do the following.

1. Select **Tools > Check Names**.
2. In the **Check Name Administration** dialog box, select the value you want to delete and either right-click it and select **Delete** or click  .
3. In the dialog box, click **Yes** to confirm that you want to delete the name.

The name is deleted, and all its associated values are deleted; the name is removed from the Check Name Administration dialog box.

Importing Names

Note: You cannot import names in tablet view.

To import names form an XML file, complete the following steps.

1. Select **Tools > Check Names**.
2. In the **Check Name Administration** dialog box, click  and select **Import**.
3. In the **Import Check Names** dialog box, click **Browse** and navigate to the file that you want to import.

4. Click **OK**.

Exporting Names

Note You cannot export names in tablet view. To export names to an XML file, do the following.

1. Select **Tools > Check Names**.
2. In the **Check Name Administration** dialog box, click  and select **Export**.
3. Save the XML file in the appropriate location.

Viewing Log Messages

You can view log messages that provide information about Perceptive DataTransfer jobs.

1. Perform one of the following actions.
   - In the Job dashboard, select a job and click the **Log** button.
   - In the Job dashboard, right-click a job and select **Log**.
   - In the Job Dashboard, select a job in the table and either right-click it and select **Open** or click the
   - **Open** button; then, in the window that appears, click **Log**.
   - Select **Tools > Logs**. The Log View dialog box appears and displays the following columns of information.
     - Job Name: Name of the job.
     - Type: Type of log message (error, SQL). To log SQL messages, check the Log SQL check box in the Add Job dialog box for the appropriate job; refer to Adding new jobs.
     - Message: Body of the log message.
     - User: Username of the user who was logged in.
     - Date/Time: Date and time on which the message was logged.
2. To filter messages by job, select the job from the **Job** drop-down box.
3. To filter messages by type, select the type of log message from the **Type** drop-down box.
4. To export messages in XML format, click  and then save the file in the appropriate location.
5. To view the suspend reason for suspended records, click the log record that indicates that a record status changed to **Suspended** and then click  to open the **Detailed Log View** dialog box, which displays the suspend reason.

Configuring Data Exports

You can use data exports to create SQL queries to obtain content from any database configured for Perceptive DataTransfer. You can save the results and export them to a delimited file.

You can share data exports with other workspaces, and you can import data exports from and export data exports to an XML file.

Refer to the following topics for more information.

- Understanding Data Export Icons
- Adding Data Exports
- Configuring and Modifying Data Exports
- Copying Data Exports
- Deleting Data Exports
- Running Data Exports
- Extracting Data Exports
- Removing Sharing on Shared Data Exports
- Rolling Back Data Exports
- Importing Data Exports
- Exporting Data Exports
- Configuring Scheduled Data Export Rules

Understanding Data Export Icons

The following table describes data export icons and their descriptions.

Icon	Description
	Data export is shared
	Data export is shared and has been modified.
	Data export is not shared and has been modified.
	Date export has not been modified. If you are working in a workspace version in edit mode, the data export is not shared. If you are working in a workspace version in a mode other than edit mode, this icon displays for all data exports. The sharing status is unknown.

Adding Data Exports

To add and configure a data export, do the following.

1. Select **Tools > Data Exports > Configure**.
2. In the **Data Exports Administration** dialog box, click  .
3. In the **Add Data Export** dialog box, perform one of the following actions.
   - To add a new data export, do the following.
     7. Click the **Add Data Export** tab.
     8. Type a name in the **Name** field and click **OK**.
   - To add a data export that is shared by another workspace, do the following.
     1. Click the **Add Shared Data Export** tab.
     2. From the **Workspace** drop-down box, select the workspace that shares the data export you want to add.
     3. From the **Data Export** drop-down box, select the shared data export that you want to add.
     4. Click **OK**.

The data export is added and appears in the Data Export Administration dialog box.

Configuring and Modifying Data Exports

1. Select **Tools > Data Exports > Configure**.
2. In the **Data Exports Administration** dialog box, either right-click the data export you want to configure and select **Edit** or click  .
3. In the **Edit Data Export** dialog box, to rename the data export, type the name in the **Name** field.
4. From the **Connection Group** drop-down box, select the connection group to use.
5. From the **Connection** drop-down box, select the database connection to use.
6. In the **Data Extract SQL** field, type the appropriate SQL code for data extract.
7. Define the format of the delimited file that contains the data returned by the query. Click the **Tab** radio button to specify that the file is tab-delimited, or click the **Specify** tab to provide your own value and then type the value in the corresponding field.
8. Check the **Include Date and Time** check box to include the date and time in the name of the data export file.
10. Click **OK**.

Copy Data Exports

To create a duplicate of a data export, do the following.

1. Select **Tools > Data Exports > Configure**.
2. In the **Data Exports Administrator** dialog box, select the data export you want to copy and click  .

The data export is duplicated, and its copy appears in the Data Export Administration dialog box.

Deleting Data Exports

To create a duplicate of a data export, do the following.

1. Select **Tools > Data Exports > Configure**.
2. In the **Data Exports Administrator** dialog box, select the data export you want to copy and click .

The data export is duplicated, and its copy appears in the Data Export Administration dialog box.

Running Data Exports

To run the query and view the data returned from the database, do the following.

1. Select **Tools > Data Exports > Configure.**
2. In the **Data Exports Administrator** dialog box, select the data export you want to run.
3. Click .
4. Click .

11. One of the following occurs.

- If there are issues with the SQL code, a dialog box appears and displays information about the problem.
- If there are no issues with the SQL code, results of the data export are displayed in the Results pane.

Extracting Data Exports

You can run the query you created and export the data returned to a file-delimited format you specified in the Edit Data Export dialog box (refer to Configuring and Modifying Data Exports).

Note You cannot extract data exports in tablet view.

1. Select **Tools > Data Exports > Configure**.
2. In the **Data Exports Administrator** dialog box, select the data export you want to extract and either right-click it and select **Extract** or click .
3. In the **Extract Data Export** dialog box, complete the following substeps.
    1. From the **Connection Group** drop-down box, select the connection group you want to use.
    2. From the **Connection** drop-down box, select the connection you want to use.
    3. Click **OK**.
    4. Save the file with the extracted data in the appropriate location.

Removing Sharing on Shared Data Exports

To remove sharing on data exports that are shared with another workspace, do the following.

1. Select **Tools > Data Exports > Configure.**
2. In the **Data Exports Administrator** dialog box, select the date export for which you want to remove sharing and click .

Sharing is removed on the data export.

Rolling Back Data Exports

If changes have been made and committed to a data export, you can use a previous version (*roll back*) of it.

1. Select **Tools > Data Exports > Configure.**
2. In the Data Exports Administrator dialog box select the data export you want to roll back and either right-click it and select **Rollback** or click . **The Rollback Manager** dialog box appears and provides the following information.
   - Name: Name of the data export.
   - Status: Status of the data export (*current* or *dated*).
   - Version Date: Date and time on which the version was created.
   - Created By: Username of the user who created the version.
   - Comments: Comments, if any, added by the user for the version.
3. Select the version that you want to use and click **OK**.

The previous version is loaded.

Importing Data Exports

Note You cannot import data exports in tablet view. To import a data export, do the following.

1. Select **Tools > Data Exports > Configure.**
2. In the **Data Exports Administrator** dialog box, select and select **Import**.
3. In the **Import Data Export** dialog box, click **Browse** to navigate to the location of the file you want to import, and then select it.
4. Check the **Update Existing Entities** check box to update an existing data export, with the same name as the data export you are importing, with the data export you are importing.

If you do not check this check box, and there is an existing data export with the same name as the data export you are importing, *DUPLICATE-* is appended to the start of the name of the data export you are importing; the existing data export is not updated. If a data export exists with the name DUPLICATE-*<data_export_name>*, *DUPLICATE(2)-* is appended to the start of the name. The number continues to be incremented so that all data exports have unique names.

5. Click **OK**.

The data export is imported and is displayed in the Data Export Administration dialog box.

Exporting Data Exports

Note You cannot export data exports in tablet view.

To export a data export as an XML file, do the following.

1. Select **Tools > Data Exports > Configure.**
2. In the **Data Exports Administrator** dialog box, click and select **Export**.

3. Save the file in the appropriate location.

Configuring Scheduled Data Export Rules

You can configure Perceptive DataTransfer to run data exports at specified times. You can also view the output of past data exports.

Refer to the following topics for more information.

- Adding and Modifying Scheduled Data Exports
- Deleting Scheduled Data Exports
- Viewing the Previous Output of Data Exports

Adding and Modifying Scheduled Data Exports

To add or modify a scheduled data export rule, complete the following steps.

1. Select **Tools > Data Exports > Schedule**.
2. In the **Schedule Data Exports** dialog box, perform one of the following actions.
   - To add a new scheduled data export rule, click  . The **Add Scheduled Data Export** dialog box appears.
   - To modify a scheduled data export rule, select the data export and either right-click it and select **Edit Scheduled Data Export** or click  to display the **Edit Scheduled Data Export** dialog box appears.
3. In the **Name** field, type the name of the scheduled data export rule.
4. In the **Status** field, click the **Enable** radio button to enable the rule; click **Disable** to disable it.
5. From the **Data Export** drop-down box, select the data export to which to assign the rule.
6. From the **Connection Group** drop-down box, select the **Connection Group** used for the data export.
7. From the **Connection** drop-down box, select the connection used for the connection group.
8. From the **Start Time** drop-down box, select the time on which the report is run.
9. In the **Options** pane, click the **Run in Edit Version** check box to run the rule against the edit version of the workspace (the rule is run against the edit version of the data export and connection). If you do not check this check box, the rule is run against the active version of the workspace.
10. In the **Recurrence Pattern** pane, specify when the report recurs.
    - Click the **Once** radio button to run the data export once. After the data export is run, rules for running the data export are disabled.
    - Click the **Daily** radio button to specify that the data export is run daily, and then specify the recurrence frequency.
    - Click the **Weekly** radio button to specify that the data export is run weekly, and then specify the recurrence frequency.
    - Click the **Monthly** radio button to specify that the data export is run monthly, and then specify the recurrence frequency.

11. Click the **Notifications** tab to configure notifications to be sent to recipients when certain data export actions occur. Refer to Configuring Notifications.
12. Click **OK**.

Deleting Scheduled Data Exports

To delete a scheduled data export rule, do the following.

1. Select **Tools > Data Exports > Schedule**.
2. In the **Schedule Data Exports** dialog box, select the rule you want to delete and either right-click it and select **Delete** or click . In the dialog box, click **Yes** to confirm that you want to delete the rule.

The rule is deleted and is removed from the Schedule Data Exports dialog box.

Viewing the Previous Output of Data Exports

To view the previous output of the scheduled data export, perform one of the following actions.

Note You cannot view data export output in tablet view.

- Select **Tools > Data Exports > View**.
- Select **Tools > Data Exports > Schedule**.

The Schedule Data Exports dialog box appears. Next, select the data export for which you want to view previous results and click .

The View Data Exports dialog box appears and displays a table with the following information.

- Date: Date on which the data export was created.
- Name: Name of the data export.
- Data Export Rule Name: Name of the rule that caused the scheduled report to be run.
- Created By: Username of the administrator who configured the data export.

You can filter the information in the View Data Exports dialog box by selecting the appropriate information in one or more of the following fields.

- Data Export Rule: Select the data export rule for which to filter results.
- Data Exports: Select the data export for which to filter results. Select *All* to specify all data exports.
- Date Range fields: Click the icon to open a calendar from which you can specify the start and end dates of a date range. You can choose both start and end dates or only a start or end date.

After specifying the filter information, click the **Filter** button to update the table with the selected criteria.

To view a data export, click . The data export opens in a new tab or browser window in PDF format.

To permanently delete the data export and the record of the data export instance, click

Viewing the Import History

To view the import history, do the following.

1. Select **Tools > Import History**.

   The Import History dialog box appears and displays the following columns of information.

- Name: Name of the object imported into Perceptive DataTransfer (for example, *Workspace* or
- *Data Export*).
- Type: Type of import (*Full Import* or *Partial Import*).
- Date Created: Date and time on which the object was imported.
- Created By: Username of the user who imported the object.

2. To view details about the Perceptive DataTransfer objects that were imported, select the entry in the table and either right-click it and select **Import Details** or click  .

The Import Report dialog box appears and displays information about the objects imported into Perceptive DataTransfer.

Configuring Notifications

You can configure Perceptive DataTransfer to send notifications to users when various activities occur in Perceptive DataTransfer. You can configure notifications for jobs, scheduled reports, and version milestones.

Refer to the following topics for more information.

- Configuring Email Addresses
- Configuring Notification Groups
- Configuring Job Notifications
- Configuring Report Notifications
- Configuring Workspace Version Notifications

Configuring Email Addresses

You must have the appropriate permissions to configure the SMTP server from which notifications can be sent, and you must also have the appropriate permissions to configure email addresses for users to which these notifications are sent.

Refer to Configuring Users and Privileges. To configure system email addresses, do the following.

1. Select **Administration > System Properties > System E-Mail**. The Configure System Email dialog box appears.
2. In the **Configure System Email** dialog box, in the **Email** address field, type the email address used that sends the system notifications.
3. In the **Host** field, type the hostname of the SMTP server that sends the emails.
4. In the **Port** field, type the port number of the SMTP server.
5. In the **Username** field, type the username of the SMTP server.

6. In the **Password** field, type the password of the SMTP server.
7. Click **OK**.
8. You can assign email addresses to Perceptive DataTransfer users; these are the emails addresses at which users receive emails sent from the SMTP server. To do so, do the following.

   Note The name and email address is updated for all users in the system with this username. For example, if user *GUEST* has multiple roles and therefore multiple instances in the system, the name and email address you configure for one instance of *GUEST* is used for all instances of that user.

   1. Select **Administration > User**.
   2. In the **User Administration** dialog box, in the **Navigation** pane, click the user for which you want to configure email settings.
   3. In the **Name** field, type the name of the user for the email address.
   4. In the **Email** field, type the name of the email address for the user.
   5. Click **Save**.

Configuring Notification Groups

After you have configured the system email address from which notifications are sent and the email addresses to which users receive those notifications, you can configure notifications for a variety of functions.

You can configure notification groups, which contain lists of email recipients (similar to an email distribution list), to which to send these notifications.

You must have the appropriate permissions to add and modify notification groups. Refer to Configuring Users and Privileges.

Refer to the following topics for more information.

- Adding and Modifying Notification Groups
- Deleting Notification Groups

Adding and Modifying Notification Groups

To add a notification group, do the following.

1. Select **Configuration > Notifications**.
2. In the **Notification Group Configuration** dialog box, perform one of the following actions.
   - To add a new notification group, click  to display the **Add Notification Group** dialog box.
   - To modify a group, select it and click  to display the **Edit Notification Group** dialog box appears.
3. In the **Name** field, type the name of the notification group.

4. In the **To** field, provide the email addresses of the recipients of the notifications. Perform one of the following actions.
   - Manually type the email addresses in the field, separating recipients by commas.
   - Click the **To** button to open the Select Users dialog box. Select one or more recipients and click **OK**.
5. In the **CC** field, provide the email addresses of the recipients to which to send copies of the notifications. Perform one of the following actions.
   - Manually type the email addresses in the field, separating recipients by commas.
   - Click the **CC** button to open the Select Users dialog box. Select one or more recipients and click **OK**.

Deleting Notification Groups

To delete a notification group, complete the following steps.

1. Select **Configuration > Notifications**.
2. In the **Notification Group Configuration** dialog box, select the group you want to delete and click  .

Configuring Job Notifications

You can configure notifications to be sent when a job activity occurs. Refer to the following topics for more information.

- Adding and Modifying Job Notifications
- Deleting Job Notifications

Adding and Modifying Job Notifications

To add a job notification, complete the following steps.

1. In the Job Dashboard, perform one of the following actions.
   - Select **Jobs > Add Jobs** to display the **Add Job** dialog box.
   - Select **Jobs > Recurring Jobs** to display the **Recurring Job Administration** dialog box appears.
2. Click the **Notifications** tab.
3. Perform one of the following actions.
   - To add a new notification, click  to display the **Add Job Notification** dialog box.
   - To modify a notification, select it and click  to display the **Edit Job Notification** dialog box appears.
4. From the **Action** drop-down box, select the job action that occurs for which a notification is sent. Options are *Complete* or *Processed*.

   When a job attains *Processed* or *Complete* status, the appropriate notifications are sent to the provided email addresses.

5. Select the recipients to which to send the notifications. You can either select a configured notification group (refer to Configuring Notification Groups, or you can specify recipients not in a group.
   - To select a notification group, select the appropriate group from the drop-down box. The **To** and **CC** fields automatically display the recipients configured for the notification group, and you cannot modify these fields.
   - To specify recipients not in a group, complete the following substeps.
     1. In the **To** field, provide the email addresses of the recipients of the notifications. Perform one of the following actions.
        - Manually type the email addresses in the field, separating recipients by commas.
        - Click the **To** button to open the **Select Users** dialog box. Select one or more recipients and click **OK**.
6. In the **CC** field, provide the email addresses of the recipients to which to send copies of the notifications. Perform one of the following actions.
   - Manually type the email addresses in the field, separating recipients by commas.
   - Click the **CC** button to open the **Select Users** dialog box. Select one or more recipients and click **OK**.
7. Click **OK**.

Deleting Job Notifications

To delete a job notification, complete the following steps.

1. Select **Jobs > Add Jobs**.
2. In the **Add Job** dialog box, click the **Notifications** tab.
3. Select the notification you want to delete and click  .

Configuring Report Notifications

You can configure report notifications when certain report activities occur. Refer to the following topics for more information.

- Adding and Modifying Report Notifications
- Deleting Report Notifications

Adding and Modifying Report Notifications

1. Select **Reports > Schedule**.
2. In the **Schedule Reports** dialog box, perform one of the following actions.
   - To add a new report, click  to display the **Add Scheduled Report** dialog box.
- To modify a report, select the report and either right-click it and select **Edit Report** or click  . The **Edit Scheduled Report** dialog box appears.
3. Click the **Notifications** tab.

4. Perform one of the following actions.
   - To add a new notification, click  to display the **Add Report Notification** dialog box.
   - To modify a notification, select it and click  . The **Edit Report Notification** dialog box appears.
5. From the **Action** drop-down box, select the report action that occurs for which a notification is sent. Option is *Complete*.

   When a report is *Complete*, a notification is sent to the specified recipients, and a PDF version of the report is included as an attachment in the email.
6. Select the recipients to which to send the notifications. You can either select a configured notification group (refer to Configuring Notification Groups), or you can specify recipients not in a group.
   - To select a notification group, select the appropriate group from the drop-down box. The **To** and **CC** fields automatically display the recipients configured for the notification group, and you cannot modify these fields.
   - To specify recipients not in a group, do the following.
     1. In the **To** field, provide the email addresses of the recipients of the notifications. Perform one of the following actions.
     - Manually type the email addresses in the field, separating recipients by commas.
     - Click the **To** button to open the Select Users dialog box. Select one or more recipients and click **OK**.
     2. In the **CC** field, provide the email addresses of the recipients to which to send copies of the notifications. Perform one of the following actions.
     - Manually type the email addresses in the field, separating recipients by commas.
     - Click the **CC** button to open the Select Users dialog box. Select one or more recipients and click **OK**.
7. Click **OK**.

Deleting Report Notifications

To delete a report notification, do the following.

1. Select **Reports > Schedule**.
2. In the Schedule Report dialog box, click the **Notifications** tab.
3. Select the notification you want to delete and click  .

Configuring Workspace Version Notifications

You can configure notifications to be sent when certain version activities occur. Refer to the following topics for more information.

- Adding and Modifying Workspace Version Notifications
- Deleting Workspace Version Notifications

Adding and Modifying Workspace Version Notifications

To add or modify a version notification, do the following.

1. Select **Administration > Workspace**.
2. In the **Workspace Administration** dialog box, select the workspace for which you want to configure notifications and click  . The Version Notification Configuration dialog box appears.
3. In the **Version Notification Configuration** dialog box, perform one of the following actions.
   - To add a new notification, click  to display the **Add Version Notification** dialog box.
   - To modify a notification, select it and click  . The Edit Version Notification dialog box appears.
4. From the **Action** drop-down box, select the version action that occurs for which a notification is sent. Options are **Active**, **Edit**, **Stage**, and **Test**.

   When a workspace moves into one of these statuses, a notification is sent to the specified recipient.
5. Select the recipients to which to send the notifications. You can either select a configured notification group (refer to Configuring Notification Groups), or you can specify recipients not in a group.
   - To select a notification group, select the appropriate group from the drop-down box. The **To** and **CC** fields automatically display the recipients configured for the notification group, and you cannot modify these fields.
   - To specify recipients not in a group, complete the following substeps.
     1. In the **To** field, provide the email addresses of the recipients of the notifications. Perform one of the following actions.
        - Manually type the email addresses in the field, separating recipients by commas.
        - Click the **To** button to open the **Select Users** dialog box. Select one or more recipients and click **OK**.
     2. In the **CC** field, provide the email addresses of the recipients to which to send copies of the notifications. Perform one of the following actions.
        - Manually type the email addresses in the field, separating recipients by commas.
        - Click the **CC** button to open the **Select Users** dialog box. Select one or more recipients and click **OK**.
6. Click **OK**.

Deleting Workspace Version Notifications

To delete a version notification, do the following.

1. Select **Administration > Workspace**.
2. In the **Workspace Administration** dialog box, click .
3. In the **Version Notification Configuration** dialog, select the notification you want to delete and click .

Configuring Jobs and Processing Records

Jobs contain all functions required to load data into your database. A job consists of the following.

- **Header**: Defines the input data source, mapping values from the input file to the source destination in your database.
- **Connection group**: Comprises database connections to your database.
- **Input Source**: Source from which you are importing data (for example, a spreadsheet containing test information for students). Input sources can be fixed-width, delimited, XML files, EDI files, database queries, or web service calls.
- **Note:** EDI files are supported in Perceptive DataTransfer 6.4.4 and higher.
- **Date and time**: Date and time on which the job starts (if not specified, the job is run manually).

When you run a job, Perceptive DataTransfer processes all records from the input source for the job.

Refer to the following topics for more information.

- Understanding the Perceptive DataTransfer desktop and tablet Job Dashboard
- Understanding the Perceptive DataTransfer mobile Job Dashboard
- Customizing columns
- Obtaining input data from the database
- Adding and modifying recurring jobs
- Adding new jobs
- Modifying job details
- Modifying jobs
- Specifying data displayed in the Results pane
- Opening jobs
- Running jobs
- Deleting jobs
- Canceling jobs
- Archiving jobs

Understanding the Perceptive DataTransfer desktop and table Job Dashboard

The following figure displays Perceptive DataTransfer Job Dashboard for the desktop and tablet views. Not all features are available on the tablet view.

The Job Dashboard provides the following information.

Element	Description
Status drop-down box	Filters the status of the jobs you view in the Job Dashboard table. When you select an option, the table displays jobs with the selected status. Options are: • Active: Active jobs are jobs that have not yet started (pending), jobs that have run but for which there are unprocessed records (incomplete), or jobs that have run but for which there are suspended records (processed). • Complete: Complete jobs have been run and all records have been processed as new or match; there are no suspended or unprocessed records. • Archive: Archive jobs are historical jobs that have been archived. • Canceled: Canceled jobs have been canceled by a user.
Name column	Name of the job. Click **+** to expand the job entry and view details about the job. The following fields are displayed. • File Name: Filename of the input file used by the job. • Created On: Date on which the job was created. • Created by: Username of the user who created the job. • Start Time: Date and time on which the job started to run. • End Time: Date and time on which the job ended. • In Use by: Username of the user who is running the job.
File Name column	Filename of the input file used by the job.
Scheduled Start Time column	Date and time on which the job was scheduled to start.

Element	Description
Status column	Status of the job. • Complete: The job has finished running, and there are no unprocessed or suspended records. • Incomplete: The job is not currently running and has processed and unprocessed records. • Processed: The job has finished running, but there are suspended records. • Pending: The job has not yet started. • Processing: The job is currently running. • Canceled: The job was canceled. • Archive: The job was archived.
Records Status column	Provides the following subcolumns. • Progress: Provides a status bar indicating job progress when it is being run. • Total: Number of total records for the job. • Unprocessed: Number of records that the job did not process. • Suspended: Number of suspended records. • Matched: Number of matched records. • New: Number of new records. • Locked: Number of locked records.
Header column	Name of the header used for the job.
Connection Group column	Name of the connection group for the job.

Understanding the Perceptive DataTransfer Mobile Job Dashboard

If you are using DataTransfer on phones with smaller screens or resolutions, the mobile Job Dashboard is displayed; the layout and functionality are the same for mobile Dashboards, but the overall design varies among devices.

The mobile Job Dashboard displays the Active Jobs page, which displays a list of active jobs. To filter the status of jobs displayed in the Job Dashboard, **Settings > Filter**, and then select the job status you want to display.

To view details about a job in the mobile Job Dashboard, select the appropriate job. The following details are displayed.

- Name: Name of the job.
- Status: Status of the job.
    - Active: Active jobs are jobs that have not yet started (pending), jobs that have run but for which there are unprocessed records (incomplete), or jobs that have run but for which there are suspended records (processed).

  - Complete: Complete jobs have been run and all records have been processed as new or match; there are no suspended or unprocessed records.
  - Archive: Archive jobs are historical jobs that have been archived.
  - Canceled: Canceled jobs have been canceled by a user.
- File: Filename of the input file used by the job.
- Progress: Displays a bar that shows the percentage of the job completion.
- Total: Total number of the job.
- Unprocessed: Total number of records not processed by the job.
- Matched: Number of matched records.
- New: Number of new records.
- Locked: Number of locked records.

You can also run jobs in the mobile Job Dashboard; for more information, refer to Running jobs.

Customizing columns

You can customize the columns displayed in the Job Dashboard.

Columns are automatically saved the way in which you displayed them when you last logged out of the application or when you switched roles.

To choose the columns you want to display, complete the following steps.

1. Hover your mouse over the appropriate column. A drop-down arrow appears on the right side of the column.
2. Click the arrow to open a menu that allows you to sort columns from A-Z or Z-A and also allows you to select the columns you want to display.
3. Select **Columns** and then check the columns you want to display; uncheck the columns you do not want to display.

After you log out of the application or change roles, the columns you specified to show or hide are displayed in the Job Dashboard the next time you log in.

Obtaining input data from the database

Perceptive DataTransfer can query a database to obtain the input data for a job. To configure a job in this way, complete the following steps.

1. Create and configure a database connection for the input data. Refer to Configuring connection groups and database connectors.
2. Create a procedure that uses Database Population Selection logic, which contains the query for dynamically extracting input data. You use this logic in procedures and associate them with headers so that you can capture header input variables from the input file.

   For more information about configuring procedures, refer to Configuring Procedures. For information about syntax for database population selection logic, refer to Understanding database population selection syntax.
3. Create a database header and select the procedure used for database population selection. Refer to Configuring and modifying database headers.

4. Create a new job, selecting the header and connection you created in previous steps. Refer to Adding new jobs.

   Note If job is recurring, the data is loaded when the job starts. If this a job you are running a single time, the data is loaded at creation time.

5. Run the job. Refer to Running jobs.

   - If data from the import source has not been loaded, it is loaded when the job runs.
   - If procedure aliases do not match the input columns (input variables in the header) in the header, the job is not run properly.
   - For recurring jobs, if a previous occurrence of a job is running, you cannot create a new job.

Adding and modifying recurring jobs

Recurring jobs run at a specified interval, using a file from a specified data source or file source.

Note For an XML or EDI header, if you change the record identifier after you create a job using the XML or EDI header, the job will not run.

Note EDI headers are supported in Perceptive DataTransfer 6.4.4 and higher.

Use file sources to configure the location of the input file; refer to Configuring File Sources. To add or modify a recurring job, complete the following steps.

1. Select **Jobs > Recurring Jobs** to display the **Recurring Job Administration** dialog box.
2. Perform one of the following actions.
   - To add a new job, click  to display the **Add Recurring Job** dialog box appears.
   - To modify a job, select the recurring job and click  . The **Edit Recurring Job** dialog box appears.
3. In the **Job Name** field, type the name of the job.
4. In the Status field, click the Enable option to enable the recurring job; click Disable to disable it.
5. From the **Header** drop-down box, select the header file used for the job.
6. From the **Connection Group** drop-down box, select the connection group.
7. From the **File Source** drop-down box, select the file source (server or SFTP connection) that contains the input file.

   File Sources are configured in Perceptive DataTransfer in Configuration > File Sources; for more information, refer to Configuring File Sources.

   This field is unavailable if you are using a database, web service, or Hobsons Connect CRM header.
8. From the **Start time** drop-down box, select the time on which the job starts. If you do not specify a start date and start time, the job is run only manually.
9. In the **Options** pane, specify the following information.
   - Check the **Log SQL** check box to log SQL messages; these messages appear in the Log View dialog box (refer to Viewing Log Messages).
   - Check the **Allow Duplicate Filenames** check box to allow jobs to have the same name.
   - Check the **Use Filename** check box to include the name of the input file in the job name. When you add a new job, the name of the input file is appended to the job name.

- Check the **Delete Files** check box to delete input files from the server after they have been used for the recurring job.

10. In the **Job Restrictions** pane, complete the following substeps.
    - Check the **Limit other users from running** check box to prevent other users from running this job.
    - Check the **Limit other users from opening or editing until processed or completed** check box to prevent other users from opening or editing the job until it has finished processing records or has completed running.
11. In the **Recurrence Pattern** pane, specify when the job recurs.
    - Click the **Daily** option to specify that the job recurs daily, and then specify the recurrence frequency.
    - Click the **Weekly** option to specify that the job recurs weekly, and then specify the recurrence frequency.
    - Click the **Monthly** option to specify that the job recurs monthly, and then specify the recurrence frequency.
    - Click the **Auto** option to specify that the job recurs automatically whenever a file is added to the file source used for the job. The Delete Files option is automatically selected to prevent duplicate jobs from being created using the same file. You can use only headers that use file sources to provide data; therefore, you cannot create automatically recurring jobs that use headers of types database, Hobsons Connect CRM, or web service.
    - Click the **Manual** option to specify that the job recurs manually. It does not have a scheduled run date and recurs only when manually started. You can start the job anytime in the Recurring Job Administration dialog box or by calling the StartManualJob web service from a third-party application (refer to *Perceptive DataTransfer API Guide* for more information). Refer to Running manual recurring jobs for information on running manual recurring jobs.
12. Click the **Notifications** tab to configure email notifications to be sent when certain job activities occur. Refer to Configuring Notifications.
13. Click **OK**.

    The recurring job is added and appears in the **Recurring Job Administration** dialog box. When an instance of this job is scheduled, the instance is created and appears in the **Job Dashboard**.

Adding new jobs

Note: For an XML or EDI header, if you change the record identifier after you create a job using the XML or EDI header, the job will not run.

Note: EDI headers are supported in Perceptive DataTransfer 6.4.4 and higher. To add a new job, complete the following steps.

1. Select **Jobs > Add Job**.
2. In the **Add Job** dialog box, the **Job Name** field displays the name of the job. The default name of the job is *USER DATE TIME*, where *USER* is the username of the user who created the job, *DATE* is the date (month, day, and year) on which the job was created, and *TIME* is the time that the job was created.
3. From the **Header** drop-down box, select the header file used for the job.

4. From the **Connection Group** drop-down box, select the connection group.
5. In the **Input File** field, click **Browse** and select the input file, which contains the data that you want to import, to use for the job.
6. From the **Dependent On** drop-down box, select the job on which you want the new job to be dependent. Jobs that you can select have one of the following statuses: *Pending*, *Incomplete*, *Connection_Error*, *Record_Key_Error*, and *On Hold*.
7. In the **Release On** field, click the appropriate option for when you want the On Hold status to be removed from the new job. If you click **Complete**, the new job starts when the job on which it is dependent obtains *Complete* status. If you click **Processed**, the new job starts when the job on which it is dependent obtains *Processed* status.
8. In the **Start Date** field, click the calendar icon to display a pop-up calendar; then, select the date on which the job starts.
9. From the **Start time** drop-down box, select the time on which the job starts. If you do not specify a start date and start time, the job is run only manually.
10. Check the **Log SQL** check box to log SQL messages; these messages appear in the **Log View** dialog box (refer to Viewing Log Messages).
11. In the **Job Restrictions** pane, complete one of the following actions.
    - Check the **Limit other users from running** check box to prevent other users from running this job.
    - Check the **Limit other users from opening or editing until processed or completed** check box to prevent other users from opening or editing the job until it has finished processing records or has completed running.
12. Click **OK**.

    The job is created and is added to the table.

    Note If you enabled **Preferences > Filename in Job Name**, the name of the input file is appended to the job name.

Modify job details

To modify job details, complete the following steps.

1. Complete one of the following actions.
    - Select a job in the Job Dashboard and right-click it and select **Details**.
    - Select a job in the Job Dashboard and click the **Details** button. The Edit Job Details dialog box appears.
2. In the **Job Name** field, type the name of the job.

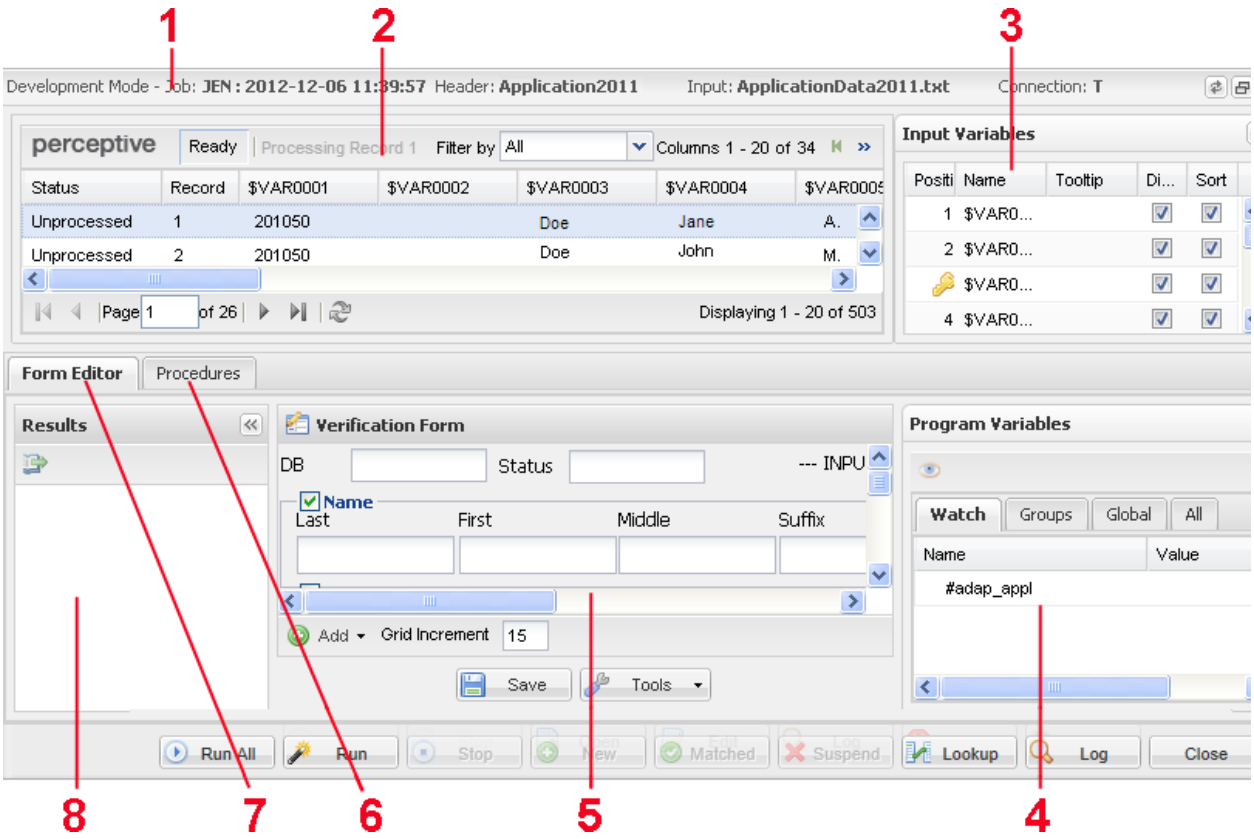
    Note You cannot modify any other fields, except the **Job Name** field, if you are modifying a job with any status other than *Pending*.
3. From the **Connection Group** drop-down box, select the connection group to use for the job.
4. In the **Start Date** field, click the calendar icon to open a calendar, and then select the date on which the job starts.
5. From the **Start Time** drop-down box, select the time on which the job starts.

6. Click **OK** to save your changes and close the dialog box.

Modifying jobs

To modify a job, select a job in the table and click **Edit**.

Note You cannot modify jobs in tablet view; you can modify them in desktop view only. A window appears and displays information about the job.



Callout No.	Element	Description
1	Menu bar	Displays the name of the job, name of the header used for the job, name of the input file used for the job, and the database connection for the job.
2	Records pane	Displays records contained in the job. You can filter the records displayed by right-clicking a record, selecting **Filter By**, and then selecting the appropriate option. You can also filter records by selecting the filter option from the Filter by drop-down box. Options are: All Unprocessed Suspended

Callout No.	Element	Description
		Match New Locked The rank column identifies the stage of matching during which the record was set as Match or New. If sorting is enabled on a column, when you click a column heading, the field is sorted alphanumerically from A-Z; if you click the heading again, the field is sorted from Z-A. You can choose to make up to ten columns sortable when you edit the input variables corresponding to the columns in the **Header Editor** dialog box. Refer to the following topics for more information. • Configuring and modifying fixed-width headers and header variables • Configuring and modifying delimited headers and header variables • Configuring and modifying database headers • Configuring and modifying Microsoft Excel headers and header variables • Configuring and modifying XML headers and header variables • Configuring and modifying EDI headers and header variables • Configuring Hobsons Connect CRM headers and header variables • Configuring and modifying web service headers Sorting is based on the prefix of the input variable. If the prefix is &, sorting is based on timestamp (which in the end is a numeric sort); if the prefix is $, sorting is alphanumeric; and if the prefix is #, sorting is numeric. These values are all stored as strings regardless of the prefix the values are converted to when the user opens or edits a job to prepare a job. If, for example, the prefix is # and the value does not convert to a numeric value, the value is managed as *null*; for #ZIP and value SEVEN, seven is *null* when sorted. You can suspend, lock, reset, and modify records by right-clicking a record and selecting the appropriate option. Refer to the following topics for more information. Viewing logs Locking records Modifying records Suspending records
3	Input Variables pane	Displays the header input variables used in the header for the job. For more information about configuring headers and header variables, refer to Configuring headers Configuringon.
4	Program Variables pane	Allows you to view and configure program variables. For more information about configuring variables, refer to Configuring Procedures.

Callout No.	Element	Description
5	Verification form	Verification form used for the job; it is displayed when you click the Form Editor tab.
6	Procedures tab and pane	Allows you to view and configure procedures. When you click this tab, the Procedures pane displays the configured procedures, and it displays the Procedure Editor pane to the right of the tab. For more information about procedures, refer to Configuring Procedures.
7	Form Editor tab	Allows you to view and configure the verification form used for the job. When you click this tab, the Verification Form pane appears to the right of the tab. For more information about configuring forms, refer to Configuring Forms.
8	Results pane	Displays any potential records in the database for which there is a match in the input record. You must configure the appropriate Match Result Display query to retrieve the appropriate information to display it in this pane. For more information, refer to Specifying data displayed in Results pane.

Specifying data displayed in the Results pane

When a job is run in interactive mode on all records, or when records that were suspended when a job was run in batch mode and are then managed in interactive mode, match logic is run and finds possible matches for the input record.

For each possible match, an entry in the **Results** pane in the **Job** window is associated with the ID for the possible match. You configure a query for a procedure using the Match Result Display logic that uses this ID to populate the Results pane in the Job window.

To configure a job that uses the Match Result Display logic, complete the following steps.

1. Create and configure a database connection for the input data. Refer to Configuring connection groups and database connectors.
2. Create a procedure that uses Match Result Display logic, which contains the query that returns information to the Results pane based on the specified ID returned by the query.
3. For more information about configuring procedures, refer to Configuring Procedures.
4. Create a header and select the procedure used for match result display. Refer to Configuring and modifying database headers.

   When the Match logic executes on a header, a list of IDs of possible matches are generated. The Match Result Display query for that header is then executed for each ID and the data returned from the query is displayed as a row of information in the Results pane.
5. Create a new job, selecting the header and connection you created in previous steps. Refer to Adding new jobs.
6. Run the job. Refer to Running jobs.

   When the job is run in interactive mode, all possible matches are displayed in the Results pane.

   Note You cannot run jobs in interactive mode in tablet view.

Opening jobs

To open a job, complete the following steps.

1. Complete one of the following actions.
   - Select a job in the table in the **Job Dashboard** and click the **Open** button.
   - Double-click a job in the **Job Dashboard** table.

     A window appears and displays information about the job.

     Note Only the **Records** pane appears in tablet view
2. To filter records in the **Records** pane, right-click a job, select **Filter By**, and select the status of the records you want to display. You can also select the filter option from the **Filter by** drop-down box. Options are:
   - All
   - Unprocessed
   - Suspended
   - Match
   - New
   - Locked

   Records text is colored according to status.
   - Locked: Purple
   - Match: Green.
   - New: Blue
   - Suspend: Red
   - Unprocessed: Black

   The Rank column identifies the stage of matching during which the record was set as Match or New. The value indicates the level of match routines executed before you determined that the record was Match or New. If the record is unprocessed or suspended, the rank value is empty.

   For more information, refer to Understanding record rank.

Status	Record	$TERM
Locked	1	201050
Unprocessed	2	201050
New	3	201050
Match	4	201050
Suspended	5	201050

3. If sorting is enabled on a column, when you click a column heading, the field is sorted, depending on the type of information contained in the column. The field can be sorted alphanumerically, numerically, or by date.
4. In this window, you can also complete the following steps.
   - Reset a record.
   - Lock a record.
   - Modify a record.
   - Suspend a record.
   - View the reason that a record was suspended.
   - View XML records (desktop view only). Refer to Modifying records.

Resetting records

To reset any *Processed* record to *Unprocessed*, complete the following steps.

1. Select a job in the **Job Dashboard** and click **Open**. A window appears and displays information about the job.
2. Complete one of the following actions.
   - In desktop view, right-click the record in the **Records** pane and select **Reset Record**.
   - In tablet view, select the record and click the **Reset** button.

   The record is reset to *Unprocessed* and its information is updated in the **Records** pane.

Locking records

When you lock a record, it is not processed when you run a job. To lock a record, complete the following steps.

1. Select a job in the **Job Dashboard** and click **Open Job**. A window appears and displays information about the job.
2. Complete one of the following actions.
   - In desktop view, right-click the record in the **Records** pane and select **Lock Record**.
   - In tablet view, select the record and click the **Lock** button.

The record is locked. Its status is changed to *Locked* in the **Records** pane.

Modifying records

To modify the value of a field of information, complete the following steps.

1. Select a job in the Job Dashboard and click Open Job. A window appears and displays information about the job.
2. Complete one of the following actions.
   - In desktop view, right-click the record in the **Records** pane and select **Edit Record**. The **Record Editor** dialog box appears.
   - In tablet view, select the record and click the **Edit** button. The **Record Editor** dialog box dialog box appears.
   - To view XML records, in desktop view, right-click a record (for a job using an XML or EDI header) and select **View Full Record**.

     Note: EDI headers are supported in Perceptive DataTransfer 6.4.4 and higher.
3. Complete one of the following actions.
   - If you are modifying records for jobs using headers other than XML or EDI headers, the **Record Editor** dialog box appears and displays the following information about the variables for the record.
     - **Pos**: Numerical value of the position of the field in the input file.
     - **Field Name**: Name of the field.
     - **Value**: Value of the field.

       Note: EDI headers are supported in Perceptive DataTransfer 6.4.4 and higher.
   - If you are modifying records for jobs using XML or EDI headers, the **Edit XML Record** dialog box appears. In the **Node** column, a icon appears next to XML tags that will appear in the **Records** pane; a icon appears next to XML tags that will not appear.

     To expand all nodes in the XML tree, click . To collapse them, click .

     Note: EDI headers are supported in Perceptive DataTransfer 6.4.4 and higher.
4. To select the next record, click the **Next** button; to select the previous record, click the **Previous** button.
5. To modify the value of a record, double-click the entry in the **Value** column and then type the appropriate value. For XML records, you can assign values to only nodes that do not have child nodes.
6. To apply your changes, click **Apply**. If you click **Previous**, **Next**, or **Close** without saving your changes, a dialog box appears and prompts you to confirm that you want to save your changes before the action is performed for the button you clicked.

The input record is updated with the changes you made, and its information is updated in the Records pane.

Viewing logs

Click the **View Logs** button to view information about the job. Refer to Viewing Log Messages for more information.

Suspending records

If Perceptive DataTransfer cannot upload a new record to the database when you run a job, the record is automatically suspended.

You can manually suspend a record if there is an issue with the record that you need to investigate further.

1. Select a job in the **Job Dashboard** and click **Open Job**. A window appears and displays information about the job.
2. Complete one of the following actions.
   - In desktop view, right-click the record in the **Records** pane and select **Suspend Record**.
   - In tablet view, select the record and click the **Suspend** button.
3. In the **Suspend Record** dialog box, in the **Enter suspend reason** field, type the reason for which you are suspending the record.
4. Click **OK**.

The record is suspended. Its status changes to *Suspended*, and the record text is highlighted in red in the **Records** pane.

Viewing the suspend reason

To view the suspend reason for a single suspended record, complete the following steps.

1. Select a job in the **Job Dashboard** and click **Open Job**. A window appears and displays information about the job.
2. Complete one of the following actions.
   - In desktop view, right-click the record in the **Records** pane and select **View Suspend Reason**.
   - In tablet view, select the record and click the **View Reason** button.

The **View Suspend Reason** dialog box appears and displays the reason that the record was suspended.

Searching records

You can search for information within the entire job or within a specific column of the **Records** pane.

Note Searches are not case sensitive.

To search for information in the entire job, in the **Records** pane, type the text for which you want to search in the **Find** field; then, press **Enter**.

If one or more results are returned, the **Records** pane is refreshed and all matching terms are highlighted in yellow. If more than one result is returned, the  (previous) and  (next) buttons are enabled, which you can use to navigate to any available previous or next results. When you hover your mouse over these buttons, tooltips appear and indicate the result to which you will navigate if you click the button.

To stop viewing search results, click  . The text in the **Find** field is cleared, the search controls are disabled, and the current page of records is refreshed without the highlighted terms.

You can also search for information within a specific column. The tooltip that appears when you hover your mouse over the Find field provides more information and an example of how to perform the search.

Type **[COLUMN]* at the end of the search term to search for the term only within the specific column.

For example, for the search term *M*[GENDER*], Perceptive DataTransfer searches for the term M within only the $GENDER column.

If you do not provide the correct name of the column, and the incorrect column name does not exist, Perceptive DataTransfer searches the entire job for the search terms.

For example, for *M*[$GENDE]*, if the $GENDE column does not exist, Perceptive DataTransfer searches the entire job for *M*[$GENDE]*.

Running jobs

Refer to the following topics for more information.

- Eliminating duplicate records
- Understanding run modes
- Running debug modes
- Running manual recurring jobs
- Running jobs in batch mode
- Running jobs in interactive mode
- Setting On Hold jobs
- Monitoring jobs

Eliminating duplicate records

To prevent duplicate records when a header is used to run a job, you must set a column of the header as the record key. Because batch processing processes multiple records at one time for improved performance, the header record key is needed to ensure that possible duplicates in the input source are not processed at the same time.

Selecting a header key allows Perceptive DataTransfer to process any records with different values in the selected column at the same time. You do not need to select a column that guarantees unique values, but the column should capture duplicate records. For example, if you select *Last Name* as the key to a header and process a file that has many records with the last name *Smith*, which represent different students, you prevent any two records with the same last name from being processed at the same time.

To mark a header column as the record key, complete the following steps.

1. Select **Configuration > Headers**.
2. In the **Header Administration** dialog box, select the header you want to modify and click  .
3. In the **Header Editor** dialog box, to mark a column as the record key, select the column and click  .

4. The  icon appears next to the column to denote that it is the record key.

   You can mark one column only as the record key; you cannot configure composite keys.

   Note A column that is set as the record key is also a sortable column and count toward the ten columns that can be sortable.

5. Configure other header properties, as appropriate.

   For more information about configuring headers, refer to Configuring headers.

Understanding run modes

You can run jobs in interactive or batch mode, which is determined in the associated match procedures using the appropriate global program variables. In tablet view, you cannot run jobs in interactive mode.

Perceptive DataTransfer sets the global variable GL_RUNMODE to BATCH when the job is run from the **Job Dashboard**. Note that if the associated code has interactive elements in it (for example, the STOP command does not work in batch mode), and a job is run in batch mode, error messages are received. You can configure the code to branch to specify the appropriate behavior when a job is run in either mode. For example, you can use an if...else statement in this syntax:

```
if _GL.RUNMODE_ = _GL.INTERACTIVE_
   <procedures for interactive mode>
else
   <procedures for batch mode>
end-if
```

For more information about global variables, refer to Understanding global variables in Programming Concepts. For more information about configuring procedures, refer to Configuring Procedures.

Understanding batch mode

In batch mode, a job is run without user intervention. For each input record, the following occurs.

1. Match logic is run until it completes. The logic searches for possible matches until a SKIP command is reached. The SKIP command sets the status of the input record and processes it appropriately.
   - If the status of the record is MATCH or NEW, upload logic is run.
   - If the status is SUSPEND, match logic is run for the next input record.

   Note Review logic is not run in batch mode.

2. Upload logic is run; the global status of the input record is used to either create a new record or to update an existing record.

Understanding interactive mode

Note In tablet view, you cannot run jobs in interactive mode.

In interactive mode, the job processes each input record, pausing to allow the user to intervene. For each input record, the following occurs.

1. Match logic is run.

    1. Match logic finds possible matches for the input record. For each possible match, an entry in the Results pane is associated with the ID for the possible match. The Match Result Display procedure uses that ID value to populate the **Results** pane in the **Job** window.

    For an overview of steps required to configure the logic to display matches in the Results pane, refer to Specifying data displayed in Results pane.

    2. Match logic organizes the results based on a ranking of how likely it is to be a match. For the first instance of the match logic, the logic searches for rank 1 matches, for the second instance it searches for rank 2 matches, and so on.
    3. At the end of each rank iteration, the match logic pauses to allow user input. The rank and logic pause are established programmatically through the STORE and STOP commands.
    4. The user can select one of the match results from the **Results** pane, initiating review logic.
    5. If the user does not find a match, the user can continue to run the match logic to search for the next rank of possible matches. If the match logic does not find a possible match, the user is prompted to create a new record for the input. If accepted, the upload logic will begin. If rejected, the record is suspended, and the process starts again with the next input record.

2. Review logic is run.

    6. Review logic is run when a user clicks a match record in the **Results** pane. The user interface stops taking input.
    7. Each time the review logic updates a program variable, any fields on the verification form linked to that program variable is updated in real-time.

3. When the review logic completes, the interface accepts user input. The user can complete the following actions.

    - Click a different match result: This restarts the review logic process.
    - Update the verification form: This allows the user to override data being uploaded to the database.
    - Insert the input record as a new record: This executes the new record upload logic.
    - Update an existing record: This executes the upload logic with the selected match result specified as the record to update.
    - Suspend the input record: This flags the record as suspended and starts the iteration of the next input record.

4. Upload logic is run.
    8. The **Results** pane is locked to prevent user input.
    9. The upload logic creates the new record or updates the existing record with the appropriate parameters.
    10. The process completes and moves to the next input record.

When running in interactive mode, you can step through any of the match, review, or upload logic to debug the code. You can examine the program variables any time the logic is paused for user intervention. Refer t Running debug modesfor more information about running debug modes.

You can also use the PROMPT function to display message text to a user and allow the user to update variables when a job is run in interactive mode. For more information, refer to Understanding the prompt function.

Running debug modes

You can run debug modes when processing records.

- **Normal mode**: The logic starts and executes to completion or until a STOP command is reached. When a STOP is reached, the interface pauses for user input.
- **Debug mode**: The logic pauses at each breakpoint and each STOP command. After the logic is stopped, you can continue to the next STOP command or breakpoint, or you can proceed to the next line of code.

You can insert breakpoints anywhere to pause on that line of code. The **Procedure Editor** displays the procedure being run and identifies the line on which the job has been stopped. Breakpoints are saved after you close jobs or the **Procedure Editor**, and they are also saved when you log out and log back into DataTransfer.

Note You cannot add breakpoints to *end-select* statements.

1. Complete one of the following actions.
    - In the **Job Dashboard**, select the job you want to run and click the **Edit** button. Next, in the topmost pane, select the record you want to run, and then click the **Procedures** tab.
    - Select **Configuration > Procedures**.
1. Select the procedure that contains the line of code you first want to execute.
2. Click the line number of the line to which you want to add a breakpoint.
    - A appears next to the line number, indicating that you have added a breakpoint to the line.
2. To clear a breakpoint, complete one of the following actions.
    - To clear all breakpoints in all procedures, click  in the **Procedures** toolbar.
    - To clear all breakpoints within a specific procedure, select the procedure and click  in the toolbar of the **Procedure Editor**.
    - Click the breakpoint to remove it.
3. Click **Run**.

4. The code runs and pauses at the breakpoint. The following buttons are enabled in the **Procedure Editor**.

-  Continues running the code until it reaches the next breakpoint, the STOP command (which pauses to allow the user to create new recrods, set matched reocrds, or suspend records), or the end of the code.
-  Pauses at the next line of code after this line has been run.
  When you debug in line mode, a dialog box appears and displays the full text of the SQL command, web service payload, or SOAP web message appears.
  Click OK to close the dialog box and continue processing. Click Cancel to stop processing at the line.
-  Stops processing the record.

Running manual recurring jobs

If a recurring job can be run manually, you can run it at any time by completing the following actions.

1. Select **Jobs > Recurring Jobs**.
2. In the **Recurring Job Administration** dialog box, click  .

You can specify that the recurring job is run manually in the **Add Recurring Job** and **Edit Recurring Job** dialog boxes. Refer to Adding and modifying recurring jobs for more information.

Running jobs in batch mode

To run a job in batch mode, complete one of the following actions.

- In the desktop **Job Dashboard**, select the job you want to run and select **Run** from the popup menu or click the **Run** button.
- In the mobile **Job Dashboard**, select the job you want to run and select  .

To stop a job that is running, complete one of the following actions.

- In the desktop **Job Dashboard**, either right-click the job and select **Stop** or click the **Stop** button.
- In the mobile **Job Dashboard**, select  .

After Perceptive DataTransfer processes the job, you can open the job to view job results, including information about any suspended or unprocessed records.

Running jobs in interactive mode

Note You cannot run jobs in interactive mode in tablet view.

To run a job in interactive mode, complete the following steps.

1. Select a job in the table and click **Open Job**. A window appears and displays information about the job.
2. Complete one of the following actions.
   - To process a single record in the job, select the record in the topmost pane and click the **Run**
   - button.
   - To process all records in the job, click the **Run All** button.
   - To stop running the job, click **Stop** in the bottom toolbar.

   Depending on your configuration, you may receive the **Prompt** dialog box for a record, which displays message text and allows you to update values of information for the record.

   For example, you may receive a message that the address information is not correct, and you can update address information, such as street number, city, and state.

To modify a value, click the row of in the Value column and type the appropriate text. Then, complete one of the following actions.

- Click **OK** to apply your changes. The information is updated and jobs processing resumes.
- Click **Reset** to reset the values in the Prompt dialog box to the original values.
- Click **Cancel** to stop processing on the record. The Error Processing dialog box appears and provides information about why the record could not be processed.

If you enabled the **Display All Match Results** preference, when you run a job in interactive mode, all possible record matches are compiled before the application pauses to allow you to review the possible matches. For more information, refer to Specifying Preferences.

If you are running a job and are using the DataProviderPrompt function with this job, you may receive a **Result Selection** dialog box, which displays a prompt in the top field and values, returned by the job, below it. This dialog box appears if multiple records of information are obtained from the job; select the information you want to use and click **OK**. For more information about web services, refer to Configuring data providers.

When Perceptive DataTransfer processes records in interactive mode, one of the following occurs.

- If a record cannot be processed, a dialog box appears and displays information about why the record could not be processed. The dialog box contains the following information:
  - **Procedure Name**: Name of the programming procedure that contains the problem.
  - **Line #**: Number of the line in the procedure in which the problem occurred.
  - **Error Information**: Description of the problem in the procedure.
- If a new record could not be uploaded, the record is suspended.
- If Perceptive DataTransfer cannot find a matching record in the database, a dialog box appears and prompts you to create a new record. Click **Yes** to create the record and add it to your database; click **No** if you do not want to create a new record.

- If there are potential matches in your database, Perceptive DataTransfer pauses and prompts you to review results, which are displayed in the Results pane. Perceptive DataTransfer organizes the results based on a ranking of how likely they are to be matches to the input data.

  Click  to continue to process records to search for more possible results.

  - If Perceptive DataTransfer does not find a match, you are prompted to create a new record. Click **Yes** to create the record; the record is updated in the topmost pane, with its status changing to New and its label color changing to blue.
  - If you find a matching result and want to create a matched record, select the result in the Results panel and click the **Matched** button. The record's status is updated as *Match*, its color changes to green, and it is updated in the topmost pane.

  After Perceptive DataTransfer uploads a record, an informational dialog box appears and displays the record number that was processed and if it was "successfully matched," if it was "successfully created," or if it "failed to upload."

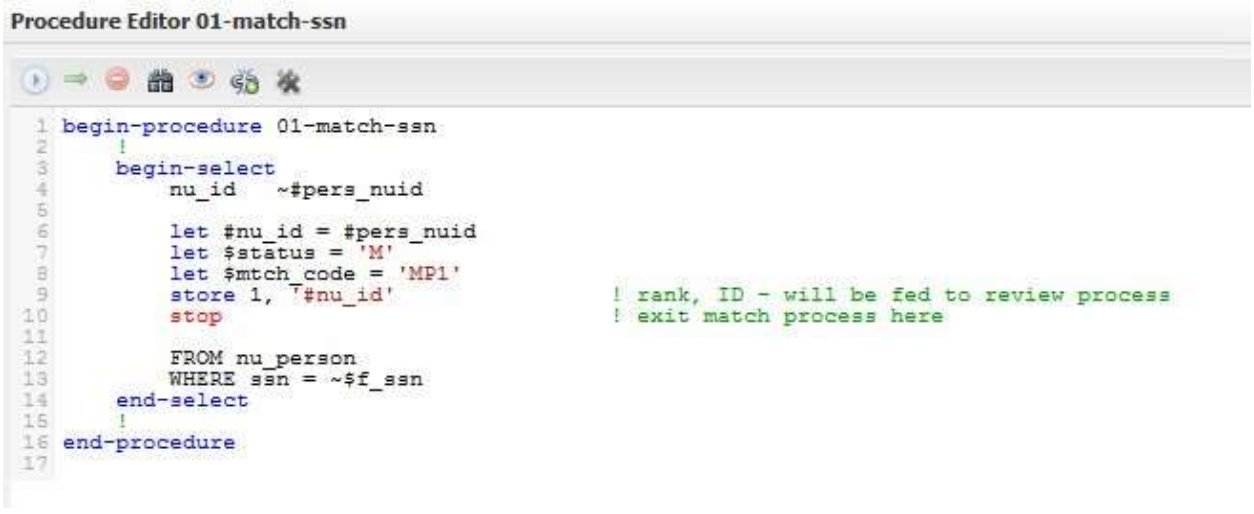
You can use the verification form, if configured, to compare information from the current input record and potential matches in your database. Refer to Viewing the verification form.

Understanding record rank

The **Rank** column in the **Records** pane in the window that appears when you edit or open jobs identifies the stage of matching during which the record was set as Match or New.

If the record is unprocessed or suspended, the **Rank** column does not display any information. If the record was set to Match or New when the job was run in interactive mode, the rank is the value used in the NQL STORE command. (For more information about the STORE command, refer to Understanding the store function).

For example, consider the example match routine in the following figure:



The match routine is *01-match-ssn* with the corresponding command *store 1, 'nu_id'*. Conventionally, match routines are numbered, and they have a corresponding store command with the same numerical value. This value is displayed in the **Match Results** pane to identify the stage of matching during which the possible match was found. Therefore, if a record is set as Match with a record with a store of *1*, the rank is also *1*. If the record is set as New, the rank is the value of the last store command. In this way, you can identify how many records the user examined before the record was set as New.

The rank process works similarly when the record is set to Match or New when the job was run in batch mode. In batch mode, the record status is changed by either a *SKIP NEW* or *SKIP MATCH* command; the *STORE* command is not used. (For more information about the SKIP command, refer to Understanding the skip function).

Both the SKIP NEW and SKIP MATCH commands take an additional second parameter, which is any string that identifies the rank at which the record status changed. The string can be any value that you want to use to indicate rank.

Consider the example match routine in the following example:

Procedure Editor AA-match-last-4-first-2-ssn

```
begin-procedure AA-match-last-4-first-2-ssn
    !
    ! 1st 4 char last, 1st 2 char first & ssn
    !
    let $where_lname_1 = 'last_idx like \q'||substr($tmp_clname,1,4)||'%\q'
    if #former_cnt = 1
        let $temp = ' OR '||'last_idx like \q'||substr($tmp_flname,1,4)||'%\q'
        let $where_lname_1 = '('||$where_lname_1||$temp||')'
    end-if
    let $where_fname_1 = 'first_idx like \q'||substr($fname_1,1,2)||'%\q'
    if #name_cnt > 3
        let $temp = ' OR '||'first_idx like \q'||substr($fname_4,1,2)||'%\q'
        let $where_fname_1 = $where_fname_1||$temp
    end-if
    if #name_cnt > 2
        let $temp = ' OR '||'first_idx like \q'||substr($fname_3,1,2)||'%\q'
        let $where_fname_1 = $where_fname_1||$temp
    end-if
    if #name_cnt > 1
        let $temp = ' OR '||'first_idx like \q'||substr($fname_2,1,2)||'%\q'
        let $where_fname_1 = '('||$where_fname_1||$temp||')'
    end-if
    !
    begin-select
    distinct(nu_names.nu_id)    ~#pers_nuid
    let #nu_id = #pers_nuid
    let $status = 'M'
    let $mtch_code = 'MPAA'
    store 1, '#nu_id'                  ! rank, UID - will be fed to review process
    !         stop                                ! have match, exit & skip to upload
    exit-select

    FROM nu_names, nu_person
    WHERE ( [$where_lname_1]
    AND   [$where_fname_1] )
    AND nu_names.nu_id = nu_person.nu_id
    AND ssn = ~$f_ssn
    end-select
    !
    evaluate _GL.ROWCOUNT_
    when = 0
        ! skip to next search
        break
    when = 1
        skip MATCH, 'AA'
        break
    when-other
        skip SUSPEND, 'Possible duplicate SSN...'
        break
    end-evaluate
    !
end-procedure
```

In this example, the match routine is *AA-match-last-4-first-2-ssn*, and its SKIP MATCH command, which has the string parameter *AA*, appears on line 45 (*skip MATCH, 'AA'*).

In this example, AA was chosen as the parameter because match procedures are numbered alphabetically; this batch procedure begins with AA. Therefore, if a record is set as Match with a record with a SKIP MATCH of *AA*, the rank is also *AA*, which indicates that the record was set to Match in match routine AA.

Similarly, if a procedure had a SKIP NEW command with a string parameter of *AA*, the record is set as

New with a rank of *AA*.

When you upgrade to this release of Perceptive DataTransfer, the rank is recorded for any new records for jobs run in interactive mode. To update record ranks for jobs run in batch mode, you must update the SKIP NEW and SKIP MATCH commands to add the new rank. This optional parameter is a string, but you can also use numeric string literals (for example, *'1'*).

Viewing the verification form

With the verification form, you can compare the current input record side-by-side with the possible existing record.

Note The form is displayed only when a job is running in interactive mode.

When you run a record, the verification form displays data associated with the input record and the possible record in your database.

Users can check or uncheck the toggle check boxes in form field sets to specify which field sets of information are included when upload logic is processed.

Use form toggles if you need to provide the value of each field set to the associated upload logic; when the upload logic is run, this value is included when the data file is uploaded. The form can return the value of a toggle given a specified program variable name.

When you configure the associated Perceptive DataTransfer logic, the logic must verify the values of the toggles and upload changed information appropriately. Refer to Using Form Toggles for more information.

After you review the data in the verification form and compare it with the possible matched record in the database, you can create a new record, create a matched record, or suspend the record.

Setting On Hold jobs

You can set a job status to be *On Hold* so that it cannot be run interactively or in batch mode until the hold status is released. You can set a job on hold if it has one of the following statuses: *Pending*, *Incomplete*, *Connection_Error*, or *Record_Key_Error*.

To set a job as on hold, select the job in the **Job Dashboard** and click **Hold**.

To remove the on hold status from the job, select the job in the **Job Dashboard** and click **Release**.

Monitoring jobs

Select a job in the **Job Dashboard** and click **Monitor**.

The Job Progress dialog box appears in the foreground of your browser and remains open and in the foreground until you click **Close**.

A progress bar appears at the top of the dialog box, indicating the percentage complete of the job. The dialog box displays the following information.

- **Total**: Total number of records in the job.
- **Unprocessed**: Number of records that are not processed.
- **Suspended**: Number of suspended records.
- **Matched**: Number of matched records.
- **New**: Number of new records.
- **Locked**: Number of locked records.

When the job is running, the **Stop** button appears in the **Job Progress** dialog box; click it to stop running the job. When the job has completed, only the **Close** button is displayed.

Deleting jobs

Refer to the following topics for more information.

- Deleting jobs
- Deleting recurring jobs

Deleting jobs

You can delete a job that has never been run (the job has a status of pending and all its records are unprocessed). You cannot delete jobs that have fully completed.

To delete a job, complete the following steps.

1. Complete one of the following actions.
   - In the desktop **Job Dashboard**, select the job you want to delete and click **Delete**.
   - In the mobile **Job Dashboard**, select the job you want to delete and click  . Perceptive DataTransfer prompts you to confirm that you want to delete the job.
1. To delete the job, click **Yes**.

The job is deleted and is removed from the **Job Dashboard**.

Deleting recurring jobs

To delete a recurring job, complete the following steps.

1. Select **Jobs > Recurring Jobs**.
2. In the **Recurring Job Administration** dialog box, select the recurring job you want to delete and click  .

3. In the dialog box, to delete the job, click **Yes**.

The job is deleted and is removed from the **Recurring Job Administration** dialog box.

Canceling jobs

You can cancel all jobs except those that has never been run (the job has a status of pending, and all its records are unprocessed).

You can cancel jobs that have at least one record that has been processed. However, you cannot cancel jobs that have fully completed.

When you cancel jobs, they are removed from the **Job Dashboard** when you select **Active** from the Status drop-down box.

To cancel a job, complete the following steps.

1. Complete one of the following actions.
   - In the desktop **Job Dashboard**, select the job you want to cancel and click **Cancel**.
   - In the mobile **Job Dashboard**, select the job you want to cancel and click  . Perceptive DataTransfer prompts you to confirm that you want to cancel the job.
2. To cancel the job, click **Yes**. The job is canceled, and its status is set to *Canceled* in the **Job Dashboard**.

Archiving jobs

You can configure rules to archive jobs in the Archive Administration dialog box. Refer to the following topics for more information.

- Adding and modifying archive rules
- Deleting archive rules

Adding and modifying archive rules

1. Select **Administration > Archive**.
2. In the **Archive Rule Administration** dialog box, complete one of the following actions.
   - To add a new archive rule, click  . The **Add Archive Rule** dialog box appears.
   - To modify an archive rule, select the rule and click  . The **Edit Archive Rule** dialog box appears.
3. In the **Status** field, click the **Enable** option to enable the archive rule; click **Disable** to disable it.
4. If you are modifying a rule for a particular header, the **Header** drop-down box displays the header; you cannot modify this field. If you are modifying the default rule for all headers, the **Header** drop-down box is not displayed.
5. In the **Days** field, type the number of days after which the job should be archived after it has completed.
6. In the **Data to Maintain** dialog box, specify the data that you want to archive.
   - **Summary Data** check box: Maintains summary data. This option is always enabled; it is unavailable to be checked.
   - **Record Level Details** check box: Check to maintain details about records.
   - **Logs** check box: Check to maintain log files.
7. From the Start Time drop-down box, select the time on which you want to start the archive rule.
8. In the **Recurrence Pattern** pane, click the appropriate option to specify whether the rule recurs daily, weekly, or monthly, and then specify the recurrence frequency.
9. Click **OK**.

If you created a new rule, it is added and appears in the **Rule Administration** dialog box. If you modified a rule, its updated information is saved.

Deleting archive rules

Note Perceptive DataTransfer includes a default, configured archive rule (which is disabled by default) that applies to all headers; you cannot delete this archive rule.

To delete a rule, complete the following steps.

1. Select **Administration > Archive**.
2. In the **Archive Rule Administration** dialog box, select the rule you want to delete and click  .

The rule is deleted and is removed from the Archive Rule Administration dialog box.

Appendix A: Programming Concepts

Refer to the following topics for more information.

- Understanding procedure syntax
- Understanding operators
- Accessing the database
- Understanding the assignment statement
- Understanding flow control
- Understanding global variables
- Understanding string functions
- Understanding general functions

Understanding procedure syntax

The body of a procedure contains the assignments, SQL statements, and flow control that comprise the procedure. After you add or modify a SQL block, you can add or modify your source code.

The syntax for the procedure body is:

```
begin-procedure procedure-name [(arg1,arg2, … argN)]
…
end-procedure
```

For example, the following procedure returns a value from a table matching on each of two input variables.

```
begin-procedure check-table-name ($t_term, $uid)
	let #found_match = 0
	let #recr_seqno = 0
	begin-select
		table-name_seqno as ~#recr_seqno

			let #found_match = 1
			exit-select


		FROM table-name
		WHERE table-name_term = ~$t_term
		AND table-name_uid = ~$uid
	end-select


end-procedure
```

In this example, *procedure-name* is the name of the procedure to be created; *arg1*, *arg2*, and so on, are the names of the procedure parameters (both input and output); and the procedure body contains the code for the procedure.

Every SQL block must begin and end with the *begin-procedure* and *end-procedure* keywords. The begin-procedure statement defines the procedure start point and the name by which the procedure is referenced and run.

Understanding database population selection syntax

Database Population Selection logic contains the query for dynamically extracting input data to a header. You use this logic in procedures and associate them with headers so that you can capture header input variables from the input file.

Example

```
begin-procedure test-query
!
       begin-select
              SSN    "$SSN",
              FIRST  "$FIRST",
              LAST   "$LAST",
              MI     "$MI",
              DOB    "+DOB",
              SEX    "$SEX",
              FROM table_name
       end-select
!
end-procedure
```

You must write database source queries in the database syntax for the target database.

Syntax for Oracle

Note the following syntax rules.

- The column alias must be enclosed by double quotation marks (“ ”)
- Use + instead of & for the column alias.

Example

```
begin-select
              SSN "#SSN",
              FIRST  "$FIRST",
              LAST   "$LAST",
              MI     "$MI",
              DOB    "+DOB"
       FROM demo
```

```
    end-select
```

Syntax for Microsoft SQL Server

Note the following syntax rules.

- The column alias must use AS and [ ]
- Use special character designators ($ for Character, # for Number, and & for Date).

Example

```
begin-select
       SSN    AS [#SSN],
       LAST   AS [$LAST],
       FIRST  AS [$FIRST],
       MI     AS [$MI],
       DOB    AS [&DOB]  FROM DEMO
end-select
```

Understanding operators

The following provides a list of symbol operators used in code.

Operator Symbol	Description
'	Character string delimiter.
+	Addition operator.
/	Division operator
-	Subtraction operator.
*	Multiplication operator.
!	Indicates a comment.
,	Separates items.
=	Equal to operator.
!=	Not equal to operator.
<=	Less than or equal to operator.
>=	Greater than or equal to operator.
<	Less than operator.
>	Greater than operator.
(	Expression or list delimiter.
)	Expression or list delimiter.
||	Concatenation operator.

Operator Symbol	Description
\q	Escape sequence for a single quote ('). When you insert strings that themselves contain strings or single quotes into the database, you must escape the single quotes. For example, to set a variable equal to the phrase "It's OK", you would use `let`

In queries, you can use the following operators.

- [ ]

Commonly, this operator is used when you are dynamically building a WHERE clause of a query. The conditions of the query depend on the code that executes and on the input data, so the WHERE clause is built inside another variable.

For example, consider a variable *$where_clause* that contains `WHERE first_name = 'John' and last_name = 'Doe'`. You can add that variable to a select statement as follows.

```
begin-select
       person_id     ~$pers_id
       FROM person
       $where_clause
 end-select
```

However, if you use the preceding query, all string variables in a query are replaced with their value surrounded by single quotes:

```
begin-select
       person_id     ~$pers_id
       FROM person
       'WHERE first_name = 'John' and last_name = 'Doe''
end-select
```

The query does not work in this manner; instead, use the [ ] operators, which tell the NQL interpreter to do a straight string replacement and to not include the surrounding quotes. Use the following syntax.

```
begin-select
       person_id
       ~$pers_id
       FROM person
       [$where_clause]
end-select
```

Example

```
begin-select
       person_id
       ~$pers_id
       FROM person
```

```
	WHERE first_name = 'John' and last_name = 'Doe'
end-select
```

- ~ (tilde)

Use the tilde operator within the SELECT clause of a query. The value of a specified column is placed into the specified variable.

Example

```
begin-select
	person_id
	~$pers_id
	FROM person
end-select
```

The value of the *person_id* column will be placed into the *$pers_id* variable. If the tilde (~) was not specified, for example:

```
begin-select
	person_id
	$pers_id
	FROM person
end-select
```

The value of the variable is inserted into the query itself. When the above query is run, no value would typically be in *$pers_i*d, and the query would look as follows.

```
begin-select
	person_id	''
	FROM person
end-select
```

However, this query syntax is incorrect. If the tilde (~) operator occurs in the FROM clause or any clause after (such as WHERE, ORDER BY, and so on) the operator is ignored. Any variable referenced in the FROM clause or after is replaced with the value in the variable at the time of execution. For the two queries:

```
begin-select
	person_id
	~$pers_id
	FROM person
	WHERE person_id = $id
end-select


begin-select
	person_id	~$pers_id
	FROM person
	WHERE person_id = ~$id
```

```
                end-select
```

If *$id = 999*, both queries evaluate to:

```
begin-select
       person_id     ~$pers_id
       FROM person
       where person_id = '999'
end-select
```

Accessing the database

You can optionally select, insert, update, and delete data from your database by embedding the appropriate SQL statements within the body of a SQL block.

Refer to the following topics for more information.

- Understanding Select constructs
- Understanding Insert, Update, and Delete constructs
- Understanding the execsp statement
- Understanding the begin-plsql statement

Understanding Select constructs

The syntax for the Select construct is:

```
[ begin-select
    [select SQL logic]
end-select ]
```

Example

```
begin-select
   last_name as     ~#tmp_last

      let #found_match = 1 exit-select
   FROM last_names
    WHERE ssn = ~$t_ssn
end-select
```

In this example, the *last_name* column is selected from the *last_names* table, where the *ssn* column matches the temporary variable *$t_ssn*.

The ~ (tilde) character is used in the SELECT clause of a query. The value of a specified column is placed into the specified variable.

Use *exit-select* to immediately exit the SQL block in which it appears. Code is run until it reaches *end-select*.

Understanding Insert, Update, and Delete constructs

The syntax for the Insert construct is:

```
begin-sql
   [insert SQL logic]
end-sql
```

The syntax for the Update construct is:

```
begin-sql
   [update SQL logic]
end-sql
```

The syntax for the Delete construct is:

```
begin-sql
   [delete SQL logic]
end-sql
```

Example

```
begin-procedure inst-names
    begin-sql
      INSERT INTO names
      SELECT #var1, $var2, $var3, sysdate,
        FROM dual
      WHERE names_id NOT IN
            (SELECT names_id
                FROM names
              WHERE names_id = ~$t_id)
       end-sql
end-procedure
```

In this example, values are inserted into a database table.

Understanding Insert, Update, and Delete constructs

The syntax for the Insert construct is:

```
begin-sql
   [insert SQL logic]
end-sql
```

The syntax for the Update construct is:

```
begin-sql
   [update SQL logic]
end-sql
```

The syntax for the Delete construct is:

```
begin-sql
   [delete SQL logic]
end-sql
```

Example

```
begin-procedure inst-names
    begin-sql
       INSERT INTO names
       SELECT #var1, $var2, $var3, sysdate,
          FROM dual
       WHERE names_id NOT IN
              (SELECT names_id
                 FROM names
               WHERE names_id = ~$t_id)
   end-sql
end-procedure
```

In this example, values are inserted into a database table.

Understanding the execps statement

This statement runs an SQL Server stored procedure.

This syntax is:

```
EXECSP spName ($param1, #param2, ~$param3…)
```

The tilde (~) is used for output parameters.

Understanding the begin-plsql statement

This statement runs an Oracle PL/SQL package or procedure.

- The begin-plsql statement allows a procedure to be called and for variables to be passed to it. It also allows the return of output variables.
- The => operator is used to assign values to variables.
- The tilde (~) character is used at the beginning of a program variable to indicate that it is an output variable type in the plsql code.

The syntax is:

```
begin-plsql
   package.procedure(
       in_variable,
       out_variable;
end-plsql
```

Example

```
begin-plsql
    nolijpackage.proc_add_name(
       first => 'John',
       last => 'Doe',
       id => ~#t_id);
end-plsql
```

Understanding the assignment statement

The *let* statement is the Perceptive DataTransfer assignment operator, which is primarily used to assign a value (for example, constant, date, or other variable) to a variable.

The syntax for the let statement is:

```
let $var1 = $var2
let $var3 = tostring(#var4)
let $var5 = &var6
```

Character and string variables can be assigned only character values. Use the *tostring* function for conversion if you are assigning a numeric value to a character variable. Refer to Understanding the tostring function.

Numeric variables can be assigned only numeric values. Use the *tonum* function for conversion if you are assigning a character value to a numeric variable. Refer to Understanding the tonum function.

You must enclose hard-coded character values in single quotation marks (') .

Example

```
let #var1 = 45 let $var2
= $var1
let $var3 = 'test string'
```

Understanding flow control

Perceptive DataTransfer supports the following flow-control statements:

- if...else: Evaluates a statement or value and branch, depending on the result. If TRUE, program execution branches to the code following the *if* statement. If FALSE, program execution branches to the code following the *else* statement. Refer to Understanding the if...else statement.
- loop...until: Allows multiple lines of program code to be executed continuously until a certain condition or value exists. Refer to Understanding the loop…until statement.
- evaluate: Evaluates a variable with multiple possible values and branches to a different block of logic for each value. Refer to Understanding the evaluate statement.
- for each: Iterates over all the specified nodes within a given subtree in an XML file. Refer to Understanding the for each statement.

Understanding the if...else statement

The if…else statement is used to evaluate a statement or value and then branch, depending on the result. There are two basic forms of the if statement: *if* and *if…else*.

The *if* statement is most frequently used to perform an action when a certain condition is met or to skip the action if the condition is not met. The syntax is:

```
if <condition>
   <statement-list>
end-if
```

If the condition in the *if* statement is TRUE, all logic contained in the statement list is run, and if the condition is FALSE, all logic contained in the statement list is skipped. The *end-if* statement terminates the statement list:

```
if #var1 > 40
   let #var3 = 45
end-if
```

The *if...else* statement uses the keyword *else* followed by a second set of statements, as follows.

```
if <condition>
   <statement-list>
else
   <statement-list>
end-if
```

If the condition is FALSE, Perceptive DataTransfer executes the statement list following the *else* clause instead of the statement list following the *if* clause:

```
if #var1 > 40
   let $var3 = 'Over 40'
else
   let $var3 = 'Under 40'
end-if
```

Additional *if* statements can be nested within each other.

```
if #var1 > 40
   let $var3 = 'Over 40'
else
   if #var1 = 40
      let $var3 = 'Equal 40'
   else
      let $var3 = 'Under 40'
   end-if
end-if
```

You can also use the *elseif* command as a shortcut for nesting statements so that they are more readable and do not require an *end-if* command for every condition, as shown in the following example.

```
if <condition 1>
elseif <condition 2>
elseif <condition 3>
....
elseif <condition N> else
end-if
```

The ellipses (...) represents that you can use any number of *elseif* conditions. If you use the *elseif* command, you must also end with an *else* clause.

Understanding the loop…until statement

The *loop…until* statement allows multiple lines of program code to be run continuously until a certain condition or value exists. The *until* statement evaluates with each loop to determine if the loop should continue.

Example

```
#c = 0
loop
   <statement-list>
    let #c = #c + 1
until #c = 5
```

This loop executes each line in the statement list and increments the *#c* variable by one until it reaches five, after which the loop is completed.

Note The *until* statement must be the last statement in the loop construct.

Understanding the evaluate statement

The *evaluate* statement is used to interpret a variable with multiple possible values and to branch to a different block of logic for each value or range of values. The evaluate statement is suitable for branching to three or more possible locations.

Example

```
evaluate #age
    when > 40
       let $display_text = 'Over 40'
       break
   when = 40
       let $display_text = 'Equal 40'
        break
   when < 40
   let $display_text = 'Under 40'
    break
end-evaluate
```

You must use the *break* statement to exit the *evaluate* statement after you have executed the appropriate code to prevent program execution from continuing with the next *when* statement.

You can use consecutive *when* statements if you are executing the same logic block. Use *when-other* as the exception handler.

Understanding the for each statement

The *for each* statement iterates over all the specified nodes within a given subtree in an XML file. Use this statement when you are processing XML input files and XML headers.

Use the *node* function within each *for each* statement to return the value for the specified node.

Example

```
for each 'Person'
   let $f_ssn = node('SSN')
   let $f_dob = cnvdate(node('Birth'),yyyy-MM-dd')
   let $f_sex = substr(node('Gender'), 1, 1)


      for each 'Name'
       let $f_first = node('FirstName')
      let $f_middle = node('MiddleName')
      let $f_last = node('LastName')
   next
```

```
 for each 'Contacts'
       for each 'Address'
           let $f_street1_1 = node('AddressLine')
          let $f_city_1 = node('City')
           let $f_st_1 = node('StateProvinceCode')
           let $f_zip_1 = node('PostalCode')
       next
       for each 'Phone'
           let $f_full_phone_1 = node('Home')
       next
         for each 'Phone'
           let $f_full_phone_2 = node('Home')
        next
    next
next
```

In this example, each record corresponds to the XML subtree contained within the <Student> XML tag. The example code shows how to process each Student subtree.

Each *for each* loop specifies the tag, exactly as it appears in the XML, with which to iterate. In this example, *for each* loops iterate over every <Person> node within the <Student> subtree.

Use the keyword *next* to designate the end of the loop.

By nesting *for each* loops, with each inner loop iterating over the child nodes of the parent, you can iterate through the entire XML tree.

Use the *node* function within each loop to return the value of the specified node. For example, the function iterates over each <Person> node, then over each <Contacts> node, and then over each <Address> node. Within the <Address> node are the leaf nodes <AddressLine>, <City>, <StateProvinceCode>, and <PostalCode>. To store the address line in a program variable, use the syntax

```
let $f_street1_1 = node('AddressLine').
```

Note the following syntax restrictions.

- If you use XML syntax (*for each* or *node*) with input record data that is not XML, an error message displays when the corresponding job is run.
- If you iterate over a node that does not exist or that does not exist as a child to the current node, the loop is skipped because no matching child nodes are found.
- If you use the node function to reference a node that does not exist, an empty string is returned.
- If you use the node function to reference a node that is not a leaf node, the function returns a concatenated string of all the children nodes for the specified node.

  For example, if you use node('Address') instead of the leaf nodes of <Address>, you may receive, for example, 138 Conant Street Beverly MA 02915 as output. For nodes such as <Address>, such output may be useful; however, for a node like ('Contacts'), you would receive a string of all the addresses and phone numbers concatenated together, which would likely not be very useful.

- Because this XML syntax allows you to reference every node in the XML by the exact XML tag name, there are no input variables for XML header jobs. Other input files have an input variable for each data field in the file, but for XML input files, you must use the for each and node syntax to drill down to the desired part of the XML tree.
- Perceptive DataTransfer keeps the current context of the XML tree. If the code is currently within a particular for each loop, any code within that loop will only process within the current subtree of the XML. All for each and node calls are contextual, so the same tag name can exist in different levels of the XML.

Understanding global variables

Global variables are predefined variables that can be referenced at any point during program execution. Perceptive DataTransfer supports the following global variables:

- _GL.BATCH_

  Run mode variable; specifies that a job is running in batch mode.

- _GL.DBNAME_

  Database variable; contains the database name or database instance name of the current database.

- _GL.EXCEPTION_

  Variable to which an exception message is assigned in the catch block of the try...catch function. Refer to Understanding the try…catch function for more information.

- _GL.FILENAME_

  Filename variable; contains the name of the current input data file.

- _GL.FILEPATH_

  File path variable; returns the destination path configured for input files that are downloaded from a file source (refer to Configuring File Sources for more information about configuring file sources).

- _GL.FILETYPE_

  File type variable; contains the extension of the current input data file.

- _GL.HEADER_

  Header variable; contains the title of the current header file (for example, *Recruit Card* or *Web Application*).

- _GL.INTERACTIVE_

  Run mode variable; specifies that a job is running in interactive mode.

- _GL.MATCH_

  Status constant; represents a matched record.

- _GL.NEW_

  Status constant; represents a new record.

- _GL.RECORDNO_

  Global variable that contains the record number of the record being processed.

- _GL.ROWCOUNT_

Row count variable; contains the number of rows returned from a begin-select or begin-dataprovider statement.

- _GL.RUNMODE_

  Run mode variable. Perceptive DataTransfer sets GL_RUNMODE to BATCH when a job is run from the Job Dashboard. You can configure the code to branch to specify the appropriate behavior when a job is run in batch or interactive mode.

Example

```
if _GL.RUNMODE_ = GL.INTERACTIVE_
    <procedures for interactive mode>
else
    <procedures for batch mode>
end-if
```

- _GL.SUSPEND_

  Status constant; represents a suspended record.

- _GL.STATUS_

  Status variable; contains the status (*new*, *match*, or *suspend*) of the row being processed.

- _GL.USERID_

  User ID variable; contains the username of the logged in user.

Understanding string functions

String functions are built-in routines for manipulating string variables and values. Refer to the following topics for more information.

- Understanding the ascii function
- Understanding the inschr function
- Understanding the instr function
- Understanding the length function
- Understanding the lower function
- Understanding the ltrim function
- Understanding the mixed function
- Understanding the mixedlt function
- Understanding the replace function
- Understanding the rtrim function
- Understanding the substr function
- Understanding the tonum function
- Understanding the tostring function

Understanding the ascii function

This function converts a character to its ASCII equivalent.

The syntax is:

```
ascii(string)
```

Example

```
let #num = ascii('X')
```

Understanding the inschr function

This function converts a character to its ASCII equivalent. The syntax is:

```
inschr(variable, replace character)
```

Example

```
let $city = inschr($city, '&')
```

This statement replaces all occurrences of *&* in the value of *$city* with the ASCII equivalent of *&*.

Understanding the instr function

The instr function searches *variable1* for the occurrence of *variable2* beginning at *start position*. The syntax is:

```
instr (variable1, variable2, start position)
```

Example

```
#pos1 = instr($street2,'Apt',1)
```

```
if instr('W|AP|AI|AF|N',$f_ethn,1) > 0
```

The function also takes an optional parameter that indicates which occurrence of the found variable should be returned. If the parameter is negative, the function searches in reverse order. If the parameter is negative and the starting position parameter is greater than one, the starting position is *n* characters from the end, counting backward.

In the following examples, you search for the / character, which occur in the first, ninth, 17th, and 25th positions.

Example

```
let $field1 = '/folder1/folder2/folder3/'
let #cnt = instr($field1, '/', 1, 1)
let #cnt = instr($field1, '/', 1, 2)
let #cnt = instr($field1, '/', 1, 3)
let #cnt = instr($field1, '/', 1, 4)
let #cnt = instr($field1, '/', 1, 5)
```

In this example, the search starts at the first character of the string and finds the nth occurrence (first, second, third, fourth, and fifth) of the / character. The first request returns *1* because the first character is the first occurrence of the / character. The second request returns *9* because the ninth character is the second occurrence of the / character. The third request returns *17*, the fourth request returns *25*; the request for the fifth occurrence returns *-1* because there is no fifth occurrence.

Example

```
let $field1 = '/folder1/folder2/folder3/'
let #cnt = instr($field1, '/', 10, 1)
let #cnt = instr($field1, '/', 10 2)
let #cnt = instr($field1, '/', 10, 3)
let #cnt = instr($field1, '/', 10, 4)
let #cnt = instr($field1, '/', 10, 5)
```

In this example, the position of the string from which the search begins is *10 (*(the second letter *o*); the first nine characters of the string are ignored. Therefore, the first occurrence of the / character is at position *17* and the second occurrence is at *25*. There are no third, fourth, or fifth occurrences of the / character.

Example

```
let $field1 = '/folder1/folder2/folder3/'
let #cnt = instr($field1, '/', 1, -1)
let #cnt = instr($field1, '/', 1, -2)
let #cnt = instr($field1, '/', 1, -3)
let #cnt = instr($field1, '/', 1, -4)
let #cnt = instr($field1, '/', 1, -5)
```

Because the last parameter (the start position) is negative, the search is performed in reverse order. In this example, the search begins at the final occurrence of the / character and works backward from the last character. Therefore, the first occurrence of the / character is at position *25*, the second occurrence is at position *17*, the third occurrence is at position *9*, and the fourth occurrence is at *1*. There is no fifth occurrence of the / character.

Example

```
let $field1 = '/folder1/folder2/folder3/'
let #cnt = instr($field1, '/', 12, -1)
let #cnt = instr($field1, '/', 12, -2)
let #cnt = instr($field1, '/', 12, -3)
let #cnt = instr($field1, '/', 12, -4)
let #cnt = instr($field1, '/', 12, -5)
```

To perform a reverse search using a start position greater than *1*, the search starts at the end of the string and moves in reverse the specified number of starting characters; then it begins searching for the nth occurrence of the character from the remaining characters.

Because the starting position is *12*, the search first counts backward twelve characters from the end. The twelve characters from reverse are *er2/folder3/*, so those characters are ignored; the remaining string is *folder1/fold*.

Next, the function begins its search backward from the last character of the remaining sting—in this example, the search begins at the final character *d* in the *folder1*/*fold* string—and finds the nth occurrence of the / character. In this example, the first occurrence of the / character is at position *9* and the second occurrence is at position *1*. There are no third, fourth, or fifth occurrences.

Understanding the length function

This function returns the length of a string. The syntax is:

```
length(string)
```

Example

```
let #num = length('Hello')
```

Understanding the lower function

This function converts a string to lower case letters. The syntax is:

```
lower(string)
```

Example

```
let $tvar = lower($t_name)
```

Understanding the ltrim function

This function removes all contiguous matching characters of type char_type from the left side of a string variable. If char_type is not specified, the ltrim function uses the space character by default. Perceptive DataTransfer begins scanning the string from the left-most character and removes all matching characters until it encounters a character of a type other than char_type.

The syntax is:

```
ltrim(string, char_type)
```

For example

```
let $unpadded = ltrim('25 Main Street', ' ')
let $unpadded = ltrim('25 Main Street')
```

Understanding the mixed function

This function converts a string to mixed case with the first characters typically being set to upper case. The syntax is:

```
mixed(string)
```

For example

```
let $street = mixed('135 washington street')
let $street = mixed('135 WASHINGTON STREET')
```

Understanding the mixedlt function

Understanding the mixedlt function

This function converts a string to mixed case, capitalizing the first character only. The syntax is:

```
mixedlt(string)
```

Example

```
let $street = mixedlt('135 vanarsdel street’)
let $street = mixedlt('135 O’HENRY STREET')
```

Understanding the replace function

This function replaces a string and takes three parameters.

- Source string
- Search string
- Replacement string

The code finds all occurrences of the search string within the source string and replaces them with the replacement string.

Example

```
let $lk-test = '12421 Main Street'
   let $lk-test = replace($lk-test, 'Street', 'St')
```

In the preceding example, the result returned by the replace function is *'12421 Main St'*.

Understanding the rtrim function

This function removes all contiguous matching characters of type *char_type* from the right side of a string variable. If *char_type* is not specified, the rtrim function uses the space character by default. Perceptive DataTransfer begins scanning the string from the right-most character and removes all matching characters until it encounters a character of a type other than char_type.

The syntax is:

```
rtrim(string, char_type)
```

Example

```
let $unpadded = rtrim('25 Main Street', ' ')
let $unpadded = rtrim('25 Main Street')
```

Understanding the substr function

This function returns a portion of a string beginning at *start_pos* and extending *char_len* characters. If *start_pos* plus *char_len* is greater than the length of string, the extra characters are ignored.

The syntax is:

```
substr(string, start_pos, char_len)
```

Example

```
let $short_var = substr('last first middle',6,12)
```

Understanding the tonum function

This function converts a string to a number.

The syntax is:

```
tonum(string)
```

Example

```
let #num = tonum('123.45')
```

Understanding the tostring function

This function converts a number to a string.

The syntax is:

```
tostring (number)
```

Example

```
let $term = tostring(#year)
```

Understanding the trunc function

This function returns a truncated version of a long date.

The syntax is:

```
trunc(date value)
```

Example

```
let $new_date = trunc($sysdate)
```

Understanding the unstring function

This function splits a string into two smaller strings depending on the occurrence of a separator character.

This syntax is:

```
unstring($string1, $delimiter, $out1, $out2)
```

Example

```
unstring 'John Doe' by ' ' into $fname and $lname
```

Understanding the upper function

This function converts a string to upper case letters.

The syntax is:

`upper(`*`string`*`)`

Example

```
let $tvar = upper($t_name)
```

Understanding general functions

General functions are a mixed collection of general purpose routines. Refer to the following topics for more information.

- Understanding the begin-dataprovider function
- Understanding the CopyFile function
- Understanding the cnvdate function
- Understanding the DataProviderCall function
- Understanding the DataProviderPrompt function
- Understanding the do function
- Understanding the ExtractBinary function
- Understanding the HobsonsUpdate function
- Understanding the ImageNowAddDoc function
- Understanding the ImageNowAddToFolder function
- Understanding the ImageNowAddToQueue function
- Understanding the ImageNowCreateFolder function
- Understanding the ImageNowUpdateNote function
- Understanding the ImageNowUpdateCustomProperty function
- Understanding the ImageNowUpdateDocumentProperty function
- Understanding the ImageNowAddStickyNote function
- Understanding the ImageNowGetFolder function
- Understanding the InitializeVars function
- Understanding the InsertBinary function
- Understanding the isnull function
- Understanding the logit function
- Understanding the lookup function
- Understanding the MoveFile function
- Understanding the NWAddDoc function

Understanding the begin-dataprovider function

You can use a begin-dataprovider/end-dataprovider block of code to run a specified data provider and iterate over its results. Use the following syntax.

```
begin-dataprovider $webServiceName
```

For example, consider the following begin-select syntax.

```
begin-select
       nu_id  ~#pers_nuid let
       #nu_id = #pers_nuid let
       $status = 'M'
       let $mtch_code = 'MP1'
       store 1, '#nu_id'    ! rank, ID - will be fed to review process
       stop   ! exit match process here
       FROM nu_person
       WHERE ssn = ~$f_ssn
   end-select
```

The preceding example could use a data provider instead of a SQL query, as shown in the following example.

```
begin-dataprovider 'personWS'
       let $status = 'M'
       let $mtch_code = 'MP1'
       store 1, '$ws_id'     ! rank, ID - will be fed to review process
       stop                  ! exit match process here
   end-dataprovider
```

After `begin-dataprovider`, you must provide the name of the data provider to run. In the preceding example, `personWS` is the name of the data provider.

The same code can be used inside `begin-select` as it is inside `begin-dataprovider`. Also, similar to `begin-select`, `begin-dataprovider` runs the data provider and iterates over the results, running the lines of codes for each result.

Like an exit-select statement that exits the begin-select/end-select code block, `exit-dataprovider` exits the begin-dataprovider/end-dataprovider code.

You can also use the optional `$payload` parameter to replace the existing payload value of the web service before it runs. Using this parameter, you can dynamically create the payload and pass it to the web service at runtime. Use the following syntax.

```
begin-dataprovider $webServiceName $payload
```

Use this function with SOAP web service or web service data providers. For more information, refer to Configuring data providers.

Note: SOAP web service data providers are supported in Perceptive DataTransfer 6.4.5 and higher.

Understanding the CopyFile function

This function copies a specified file from the source path to the destination path. It takes three parameters: the filename, the source path, and the destination path.

You can use wildcard characters in the filename parameter. If you use a wildcard, all files matching the specified criteria are moved.

Note If you a copy a file that has the same filename as a file in the destination path, the existing file is overwritten.

Example

```
CopyFile('temp.xml', 'c:/', 'c:/backup/')
```

Understanding the cnvdate function

This function converts multiple long date values into a formatted date string. This function takes two parameters: the string to convert into a date and the format of the new date string. For the first parameter, the string must be in one of the following formats to be recognized as a valid date.

- dd-MMM-yy
- MM-dd-yy
- yyyy-MM-dd
- yyyy/MM/dd
- MMM dd yyyy
- dd-MMM-yyyy
- MM-dd-yyyy
- MM/dd/yyyy

The syntax is `cnvdate(date value, 'format string')`

Example

```
let $tmp_date = cnvdate($t_date, 'DD-MMM-YYYY')
```

Understanding the DataProvider Call function

This function uses two parameters.

- The `$webServiceName` parameter is the name of the web service to run.
- The optional `$payload` parameter replaces the existing payload value of the web service before it runs. Using this parameter, you can dynamically create the payload and pass it to the web service at runtime.

The syntax is:

```
DataProviderCall ($webServiceName, $payload)
```

Example

```
let $ws_id = '10' DataProviderCall('personWS')
```

In this example, personWS is the name of the web service you run. When Perceptive DataTransfer reaches this line of code, it performs the following tasks to run the personWS web service.

1. Perceptive DataTransfer looks up the web service by the supplied data provider name.
2. It verifies the current connection group, which is the group configured for the job configured to run, for the saved connection name.
3. Perceptive DataTransfer finds the connection for the connection group. One of the following situations occurs.

- If it finds the matching connection, it uses the URL to run the web service request; then, it populates the input parameters with the values of the mapped program variables. If the web service returns more than one result, the record suspends with a *Too many results returned* reason.
- If it does not find a matching connection, the web service request results in an error.

In the preceding example, after you run the personWS web service, the output values are inserted into the mapped program variables. The value of the `$ws_fist_name` parameter is `John`, the value of the `$ws_last_name` parameter is `Doe`, and the value of the `$ws_id` parameter is `10`.

Use this function with SOAP web service or web service data providers. For more information, refer to Configuring data providers.

Note: SOAP web service data providers are supported in Perceptive DataTransfer 6.4.5 and higher.

Understanding the DataProvider Prompt function

The command runs the specified web service, as described in Understanding the DataProvider Call function. It also displays a prompt that users can use to view results.

Use the following syntax.

```
DataProviderPrompt $prompt, $webServiceName, $hidePromptWhenEmpty, $autoSelectResult,
$payload
```

Example

```
let $ws_id = ' '
let $ws_first_name = ' '
let $ws_last_name = ' '
DataProviderPrompt 'Choose the matching person.', 'personWS'
```

This function uses two required parameters: a prompt (`$prompt)` and the name of the web service (`$webServiceName`).

You can also set two optional string parameters (for example, string literals, string expressions, string variables, and so on). Use a `Y` (or `y`) value to indicate that the option is enabled. You can also use the optional `$payload` parameter. The optional parameters are listed below.

- The `$hidePromptWhenEmpty` parameter is a flag to hide the prompt if no results are returned by the data provider—if the request had no results, Perceptive DataTransfer does not prompt the user to select one.

  Example

  ```
  let $ws_id = ' '
  let $ws_first_name = ' '
  let $ws_last_name = ' '
  DataProviderPrompt 'Choose the matching person.', 'personWS', 'Y'
  ```

- The `$autoSelectResult` parameter is a flag to automatically select the result if exactly one result is returned by the data provider—if the data provider returns a single record, the record is auto-selected and no prompt displays.

  Example

  ```
  let $ws_id = ' '
  let $ws_first_name = ' '
  let $ws_last_name = ' '
  DataProviderPrompt 'Choose the matching person.', 'personWS', 'Y', 'Y'
  ```

- The optional `$payload` parameter replaces the existing payload value of the web service before it runs. Using this parameter, you can dynamically create the payload and pass it to the web service at runtime.

In the **Result Selection** dialog box, the prompt you supply appears in the field at the top of the dialog box. All results returned by the web service appear in the table below.

After a user selects a row, the **OK** button is enabled. If the user clicks **OK**, all the values from the selected row are placed into the mapped program variables.

This prompt appears when a job is running in interactive mode only. If this function is processed when a job is run in batch mode, Perceptive DataTransfer checks the number of return results. The web service performs the following actions.

- If there are no results, the output program variables are set to the empty string.
- If there is one result, the output values are placed into the mapped program variables.
- If there is more than one result, the record is suspended with a `Too may results` reason.

Use this function with SOAP web service or web service data providers. For more information, refer to Configuring data providers.

Note: SOAP web service data providers are supported in Perceptive DataTransfer 6.4.5 and higher.

Understanding the do function

This function executes a Perceptive DataTransfer procedure and optionally passes in one or more parameters.

The syntax is:

```
do procedure-name [arg1, arg2,… argN]
```

Example

```
do check-dob
```

```
do lookup-name (#ssn, $last, $first)
```

Understanding the ExtractBinary function

This function takes two parameters: a SQL query that retrieves one database BLOB value and a file type that stores the binary data. It then returns the file path to the file created from the binary data.

Example

```
let $filepath = ExtractBinary($query, 'pdf')
```

In this example, the *$query* parameter contains a SQL SELECT statement that returns one record with one column of a binary data type (for example, BLOB). Perceptive DataTransfer executes the query and creates a file with the specified extension (in this example, PDF) by writing the binary data to the file. The ExtractBinary function then returns the filepath to the file that was created.

Note: The ExtractBinary function is supported in Perceptive DataTransfer 6.4.3 and higher.

Understanding the HobsonsUpdate function

You can update Hobsons Connect CRM data using Hobsons Connect CRM web services. To do so, you create a connection group that contains a Hobsons Connect CRM connection and then you can create a Hobsons Connect CRM header. You create a job that uses the Hobsons Connect CRM header and connection and can use the appropriate NQL logic that updates Hobsons Connect CRM data when the job runs.

This syntax, which is optional, is:

```
HobsonsUpdate(<web service name>, <contact id>, <attribute name 1>, <attribute value 1>, <attribute name 2>, <attribute value 2>, ... , <attribute value n>)
```

The required parameters are:

- <*web service name*>: the name of the Hobsons web service (for example, UpdateContact).
- <*contact id*>: the ID value of the contact record to update.

The additional parameters are any number of attribute name or value pairs. Both components of the pair are strings; the first string identifies the name of the Hobsons Connect CRM attribute to update and the second string is the new value for the specified attribute.

The HobsonsUpdate function returns a string. If the update is successful, an empty string is returned. If it is not successful, a corresponding error message is returned.

An example of the syntax is:

```
let $err = HobsonsUpdate('UpdateContact', $id, 'first_name', $fname, 'last_name',
$lname, 'dob', $dob')
```

This syntax runs the *UpdateContact* web service on the contact with id *$id* and updates the *first_name* attribute to the value in *$fname*, the *last_name* attribute to the value in *$lname*, and so on.

Understanding the ImageNowAddDoc function

To upload a file to ImageNow, use the following syntax:

```
ImageNowAddDoc($filepath, $document_name, $index_map)
```

- The first parameter is an absolute path to the document you want to upload. The filename of the file in the filepath parameter can contain one or more * (asterisk characters) as wildcards. If a file matches the pattern, it is used. If no file matches the pattern or if multiple files match the pattern, the record is suspended. You cannot use the * as part of the directory name.
- The second parameter is the name to provide to the document when it uploads.
    - If this document name already exists in ImageNow, the function appends a number to the end until it finds a document name that does not exist. For example, if you try to name the document *document*, but *document* already exists, the function uses the name *document(1)*. If *document(1)* also exists, the function uses the name *document(2)*, and so on.
    - You can supply an empty document name. The function uses other criteria to determine if there is a matching document with no name. If it finds a match, the function appends the file to the existing document as a new page to that document. If a matching document is not found, the function creates a new document whose document name is the document ID.
- The third parameter is the name of an ImageNow index map (refer to Configuring ImageNow Index Maps for more information). This object links all the necessary ImageNow document properties to program variables that can be used in the code. Perceptive DataTransfer then performs a lookup on the index map name to obtain all the mapping information necessary to upload the document. All ImageNow properties are replaced with the value in the corresponding program variable.

Any time the ImageNowAddDoc syntax is reached in the code, Perceptive DataTransfer obtains the Integration Server connection in the current connection group and uploads the document using ImageNow web service APIs. For more information about Integration Server connections, refer to Adding and modifying integration server connections.

It also updates the document properties, including the custom properties, specified in the selected ImageNow index map. For more information about configuring ImageNow index maps, refer to Configuring ImageNow Index Maps.

This function also returns the document ID created for the uploaded document.

You can also set two optional parameters to convert files to TIFF and to delete the original file. The following example requests show how to set optional parameters:

```
ImageNowAddDoc('c:\temppdf\test2.pdf', $docname, $index_map, 'Y')
```

and

```
ImageNowAddDoc('c:\temppdf\test2.pdf', $docname, $index_map, 'Y', 'Y')
```

The first request contains the optional parameter to convert the file to a TIFF file before uploading. Use Y to convert the file to TIFF. Any other value, including an empty string, does not convert the file to a TIFF. In the first request, test2.pdf will be converted into a temp file called test2.tif. The temporary TIFF file will be uploaded into ImageNow and will be deleted when the upload is complete. The original test2.pdf file remains.

You can configure properties for converting TIFF to PDF in Administration > System Properties > PDF to TIFF; for more information, refer to Configuring PDF to TIFF Properties.

The second request contains the optional parameter to delete the original file. Use *Y* to delete the original file. Perceptive DataTransfer deletes files every minute. In the second request, because both optional parameters are set, test3.pdf will be converted into a TIFF, the TIFF will be uploaded to ImageNow, and then both the temporary TIFF file and the original test3.pdf file will be deleted. If the first optional flag is a value other than Y, the original test3.pdf will be uploaded to ImageNow and then deleted.

The TIFF file, if one is created, is temporary and is always deleted; however, whether the original file is deleted depends on the second optional parameter.

Note Instead of using *Y*, you can also pass in string expressions, including program variables; however, those values must be equivalent to *Y* to set the flag.

Note You can convert only PDF files to TIFF files. If the file you want to convert is in any other format, the file is not uploaded and an error message appears.

Understanding the ImageNowAddToFolder function

This function moves an existing ImageNow document into a specified ImageNow folder. It takes the ImageNow document ID and ImageNow folder name. This function requires a properly configured ImageNow integration server connection in the connection group you are using. The syntax is:

```
ImageNowAddToFolder($imagenow_docid, $imagenow_foldername)
```

Parameters are:

- $*imageNow_docid*: Document ID of the ImageNow document you are moving.
- $*imagenow_foldername*: Name of the ImageNow folder into which you are moving the document.

Understanding the ImageNowAddToQueue function

This function adds an ImageNow document to an ImageNow workflow queue. The syntax is:

```
ImageNowAddToQueue($docid, $workflowprocessName, $queueName)
```

Parameters are:

- $*docid*: Document ID of the ImageNow document you are moving.
- $*workflowprocessName*: Name of the ImageNow workflow process into which you are moving the document.

  Note: The workflow history remains unchanged when documents are added to the workflow queue.
- $*queueName*: Optional parameter representing name of the ImageNow queue into which you are moving the document.

When the code reaches this function, the Integration Server connection in the current connection group is used to execute an ImageNow web service to move the document into the queue.

Understanding the ImageNowCreateFolder function

This function creates a new folder in ImageNow. It takes a folder name, drawer name, and folder type. This function requires a properly configured ImageNow integration server connection for the connection group you are using. The syntax is:

```
ImageNowCreateFolder($foldername, $drawername, $foldertype)
```

Parameters are:

- $*foldername*: Name of the ImageNow folder you are creating
- $*drawername*: Name of the ImageNow drawer into which the folder is added
- $*foldertype*: Type of folder

Understanding the ImageNowUpdateNote function

This function appends a note to an existing document.

The syntax is:

```
ImageNowUpdateNote($docid, $note)
```

It uses the following parameters:

- $docid: Valid document ID of an existing document in Perceptive Content
- $note: The note that is appended to the document

Understanding the ImageNowUpdateCustomProperty function

This function updates the custom properties of an existing document.

Note: The date type custom property accepts values in Epoch timestamp in milliseconds.

The syntax is:

```
ImageNowUpdateCustomProperty($docid, $name, $value, $iscomposite,
$compositePropertyName)
```

The ImageNowUpdateCustomProperty function uses the following parameters:

- $docid: Valid document ID of an existing document in Perceptive Content
- $name: Valid name of the custom property of the document
- $value: Valid value of the custom property of the document. If the value is Boolean, use ‘0’ for FALSE and ‘1’ for TRUE.
- $iscomposite: Optional Parameter. This parameter signifies whether the data type of the custom property is composite.
- $compositePropertyName: This parameter signifies the name of the composite property.

  Note: You must set the $iscomposite flag to Y/N. If Y, ensure that you enter a valid value for the $compositePropertyName.

Understanding the ImageNowUpdateDocumentProperty function

This function updates the document properties of an existing document.

The syntax is:

```
ImageNowUpdateDocumentProperty($docid, $name, $value)
```

It uses the following parameters.

- $docid: Valid document ID of an existing document in Perceptive Content
- $name: Valid name of the document property of the document

- $value: Valid value of the document property of the document

Understanding the ImageNowAddStickyNote function

This function adds sticky note annotation to an existing document.

The syntax is:

```
ImageNowAddStickyNote($docid, $pageid, $filepageno, $templateid, $x, $y, $content, $queueid)
```

It uses the following parameters:

- $docid: Valid document ID of an existing document in Perceptive Content
- $pageid: Valid Page ID of an existing document in Perceptive Content
- $filepageno: The file page number of the annotation
- $templateid: The ID of the annotation template used to create this annotation
- $x: X co-ordinate of the position of the sticky note annotation on the document
- $y: Y co-ordinate of the position of the sticky note annotation on the document
- $content: The note that is added to the sticky note
- $queueid: Optional Parameter. Verifies the user's process privilege of the workflow queue that the document is in. Overrides the user's open content privilege for the drawer that the document is in.

Note: Sticky notes are rendered for TIFF files only.

Understanding the ImageNowGetFolder function

This function returns the ImageNow ID for a specified folder. It takes the folder name and searches ImageNow for the folder. If a match is found, the function returns the ID of the folder. If a match is not found, an empty string is returned. This function requires a properly configured ImageNow integration server connection for the connection group you are using.

The syntax is:

```
let $folderid = ImageNowGetFolder($foldername)
```

The $foldername parameter is the name of the ImageNow folder.

Understanding the InitializeVars function

The InitializeVars function updates all program variables in a variable map to use the specified default value.

The syntax is:

```
InitializeVars 'Variable_Map_Name'
```

where *Variable_Map_Name* is the single string parameter that represents the name of the variable map.

For example, you can create a variable map for a header, with the variable map containing all the program variables and their default values for the header, and use the InitializeVars function to set all the default values.

Understanding the InsertBinary function

Use the InsertBinary function to insert a binary file directly into the database. The syntax is:

```
InsertBinary($filepath, $query)
```

Parameters are:

- Filepath: Absolute path to the file to insert into the database.
- Query: SQL query to execute that inserts or updates the record with the binary file. The query must contain a question mark (?) for which the binary file is substituted

Example

```
InsertBinary('c:/temp/file.pdf', 'INSERT INTO file_table (id, binary_file) VALUES (1,
?)')
```

The absolute path to the file is *c:/temp/file.pdf*; the query also contains a ? symbol. Perceptive DataTransfer will execute the query and replace the ? with the binary contents of the file, which inserts the file directly into the database. You can also use an SQL UPDATE statement, instead of an INSERT statement, to add the file to an existing database record.

Understanding the isnull function

This function returns TRUE if the variable contains no valid data (NULL); otherwise, it returns FALSE. It also returns TRUE if the string variable being queried is either null or is the empty string.

The syntax is:

```
isnull(variable)
```

Example

```
if isnull($city)
let $valid = isnull($dob)
```

Understanding the logit function

This function defines a custom logging type that will produce the logit type entries in the Log View dialog box.

When Perceptive DataTransfer is set up, custom logging or text may be written to the log file; this text does not cause a record to suspend but writes a comment to the log file.

The syntax is:

```
logit 'Custom log message'
```

Example

```
logit 'No match on high school code'
logit 'Generated ID: '||$id
```

Understanding the lookup function

This function substitutes an output value for an input value and stores it in a program variable. Lookup values are defined in the lookup tool (refer to Configuring Lookup Values for more information).

```
The syntax is:
Let out_variable = lookup(in_variable, field)
```

The *$out_variable* is sent to the output value from the lookup tool. The field name is passed to the lookup() function, either through a single-quoted $Field name or custom text that you define, along with the*in_variable*.

For example, for the following fields defined in the lookup tool:

- Field: ActivityCustomMapping
- In: TestInput
- Out: TestOutput

Use the following function.

```
let $tmp_var = lookup('TestInput','ActivityCustomMapping')
```

$tmp_var is set to 'TestOutput'.

For example, for the following fields defined in the lookup tool:

- Field: $ACTIVITY
- In: TestInput
- Out: TestOutput

Use:

```
let $tmp_var = lookup('TestInput','$ACTIVITY')
```

$tmp_var is set to 'TestOutput'.

Note You do not need to use a variable name; you can use other parameters, such as activity codes or a custom name.

Note You define input and output variables only one time through the lookup tool.

Understanding the MoveFile function

This function moves a specified file from the source path to the destination path. It takes three parameters: the filename, the source path, and the destination path.

You can use wildcard characters in the filename parameter. If you use a wildcard, all files matching the specified criteria are moved.

Note If you a move a file that has the same filename as a file in the destination path, the existing file is overwritten.

Example

```
MoveFile('temp.xml', 'c:/', 'c:/backup/')
```

Understanding the NWAddDoc function

This function passes documents from Nolij Web into Perceptive DataTransfer. After Perceptive DataTransfer creates a new or match record for the document, the document is sent to Nolij Web using this Nolij Web API command.

It uses the following parameters.

- #docCode: wfdt code, determined by the filename extension by Nolij Web. This value can be *null*.
- $srcPath: Directory on the server in which this file is located.
- $fileName: Filename of the file. It can contain one or more * (asterisk characters) as wildcards. If a file matches the pattern, it is used. If no file matches the pattern, or if multiple files match the pattern, the record is suspended.
- $folderID: Folder ID. String or numeric variables can be values.
- $subfolderID: Not yet supported by Nolij Web API. String or numeric variable is accepted.
- #usercode: Nolij Web usercode for the authentication username stored for the Nolij Web connection. The value can be *null*. If *null* is passed, the API uses the default usercode for the authenticated user name.

You cannot use the Nolij Web superuser account with this function. When you configure the Nolij Web connection, you must use an account that has one or more roles assigned. Any user code you use with this function must be for a role owned by the account used in creating the Nolij Web connection.

The syntax is:

```
NWAddDoc(#docCode, $srcPath, $fileName, $folderID, $subfolderID, #usercode)
```

You can also set two optional parameters to convert files to TIFF and to delete the original file.

Example

```
nwadddoc(0, 'c:\temppdf\', 'test2.pdf', '325', '', 33, 'Y')
```

and

```
nwadddoc(0, 'c:\temppdf\', 'test3.pdf', '325', '', 44, 'Y', 'Y')
```

The first request contains the optional parameter to convert the file to a TIFF file before uploading. Use *Y* to convert the file to TIFF. Any other value, including an empty string, does not convert the file to a TIFF. In the first request, test2.pdf will be converted into a temp file called test2.tif. The temporary TIFF file will be uploaded into Nolij Web and will be deleted when the upload is complete. The original test2.pdf file will remain.

You can configure properties for converting TIFF to PDF in Administration > System Properties > PDF to TIFF; for more information, refer Configuring PDF to TIFF Properties.

The second request contains the optional parameter to delete the original file. Use *Y* to delete the original file. Perceptive DataTransfer deletes files every minute. In the second request, because both optional parameters are set, test3.pdf will be converted into a TIFF, the TIFF will be uploaded to Nolij Web, and then both the temporary TIFF file and the original test3.pdf file will be deleted. If the first optional flag is a value other than Y, the original test3.pdf will be uploaded to Nolij Web and then deleted.

The TIFF file, if one is created, is temporary and is always deleted; however, whether the original file is deleted depends on the second optional parameter.

Note Instead of using *Y*, you can also pass in string expressions, including program variables; however, those values must be equivalent to *Y* to set the flag.

Note You can convert only PDF files to TIFF files. If the file you want to convert is in any other format, the file is not uploaded and an error message appears.

Understanding the NWGetDoc function

This function downloads a specified document from Nolij Web and stores the document in a temporary space on the Perceptive DataTransfer server. It returns the string path to the temporary file, which the user can use to reference and use the downloaded file. This function requires a properly configured Nolij Web connection in the connection group you are using. The syntax is:

```
let $filepath = NwGetDoc(#docid, $folderid, #usercode)
```

Parameters are:

- #*docid*: Numeric document ID in Nolij Web.
- $*folderid*: Nolij Web folder ID for the document.
- #*usercode*: User code of the Nolij Web user who is attempting to download the document. The #usercode parameter must correspond to the user configured for the Nolij Web connection in the connection group you are using.

Understanding the NWWorkComplete function

This function starts the Nolij Web workflow process and routes folders to the appropriate inboxes. This function takes four parameters:

- Nolij Web workflow code: a string parameter that contains the Nolij Web wfma code.
- Folder Id: a string parameter that contains the Nolij Web folder ID.
- Folder name: a string parameter that contains the folder name as it appears in Nolij Web.
- User code: a numeric code parameter that contains the Nolij Web user code with which to run the Work Complete action.

Example

```
NWWorkComplete('289', '1464', 'Adams, Nicole', 44)
```

In the preceding example, *289* is the wfma code, *1464* is the folder ID for *Nicole Adams*, *'Adams, Nicole*' is how the folder is named in Nolij Web, and *44* is the Nolij Web user code with which to run the Work Complete action.

Understanding the PopulateTemplate faction

Use the PopulateTemplate function with document templates. When you create a new document template that references program variables in its text, the PopulateTemplate function applies the values of the variables to the template and creates a new text file.

Example

```
let $filepath = PopulateTemplate($template_name)
```

In this example, the PopulateTemplate function looks up the document template by the specified template name parameter, replaces all program variables in the template with their values, and then creates a new text file in a temporary location. The PopulateTemplate function returns the file path of the new text file. The format of the text file depends on the document type specified for the document template (for example, a TXT type creates a .txt file, and XML type creates an .xml file, and so on). You could then use the ImageNowAddDoc or NWAddDoc function to upload the created text file.

You can use two optional parameters—file source name and filename—with the PopulateTemplate function. You can use these parameters to specify where to place the file after running the PopulateTemplate function. If you provide a file source name and filename, the function retrieves the configured file source for the workspace version and creates the file in the path of the file source; the file created is named with the value supplied in the filename parameter.

If you supply a file source name, you must also supply the filename.

Example

```
let $filepath = PopulateTemplate('XML for Auto Import', 'ServerPath', 'temp.xml')
```

Understanding the prompt function

This function displays a string of message text to the user and allows users to update values for displayed variables. You define the prompt text to display and you define the program variables that the user can update. You also define the label that is displayed for the variable; this label is displayed to the user instead of the actual variable name.

When a job is run in interactive mode and this line of code is reached, job processing is paused, and the Prompt dialog box appears. The dialog box displays the prompt message and the labels for the variables you defined; it also displays the corresponding values for the variables. The user can modify the value of the variables.

The syntax is:

```
prompt string, variable1, label1, variable2, label2...variableN, labelN
```

Example

```
prompt 'Address is not valid. Please adjust accordingly.', $fstreet1_1, 'Street 1',
$fstreet1_2, 'Street 2', $f_city_1, 'City', $f_st_1, 'State', $f_phone_1, 'Phone'
```

Understanding the skip function

This function skips all other processing and updates the status on the record. The first parameter is the status to use: NEW, MATCH, SUSPEND, or LOCK.

The syntax within matching routines is:

```
skip [NEW/MATCH/SUSPEND], 'Reason for new/match/suspend'
```

Example

```
skip SUSPEND, 'Invalid birth date'
```

The syntax within procedure code is:

```
skip [SUSPEND/LOCK], 'Reason for suspend/lock'
```

The skip LOCK function locks the record, highlights it in a fuchsia color, and changes the status of the record to *Locked*.

The SKIP NEW and SKIP MATCH commands, using additional second parameters, can be used in code for jobs run in interactive mode to identify the rank value for a record; the rank indicates the stage of matching during which the record was set as Match or New. The rank value appears in the Rank column in the window that appears when you open or edit jobs. For more information, refer to Understanding record rank.

Understanding the stop function

This function causes program execution to pause and the Perceptive DataTransfer window to switch into review mode for any records previously collected using the store command.

This syntax is:

```
stop
```

Example

```
<statement-list>
```

```
store 1, '#uid'
stop
```

Understanding the store function

This function causes the matching set of unique IDs for the current record, in addition to a number indicating the relative matching step, to be stored internally for subsequent display and review by an operator.

The syntax is:

```
store step#, uid_var
```

Example

```
<statement-list>
store 1, '#uid'
```

In addition, the STORE command in code for jobs run in interactive mode is used to identify the rank value for a record; the rank indicates the stage of matching during which the record was set as Match or New. The rank value appears in the Rank column in the window that appears when you open or edit jobs. For more information, refer to Understanding record rank.

Understanding the toggle function

This function returns the state of a form field set toggle associated with the variable being passed in.

All variables linked to form objects are associated with a field set, which may have a visible form toggle. A form toggle is an interactive check box indicating whether a group of fields will be included in the data upload process.

Passing any variable linked to a form group into the toggle function returns the state of the toggle button for that field set as either TRUE (data in this field set will be included in the upload) or FALSE (data in this field set will not be included in the upload).

Toggle check boxes can be manually overridden to force certain data to be loaded or cause certain data to be skipped.

The syntax is:

```
toggle(variable)
```

Example

```
if toggle($suffix)
```

Understanding the toggle_off function

This function causes the form field set toggle associated with a linked form variable to be turned off (set to FALSE).

The syntax is:

```
toggle_off(variable)
```

Example

```
toggle_off($last)
```

Understanding the toggle_on function

This function causes the form field set toggle associated with a linked form variable to be turned on (set to TRUE).

The syntax is:

```
toggle_on(variable)
```

Example

```
toggle_on($last)
```

Understanding the try…catch function

This function discovers possible exceptions that could occur in a block of code and manages the exception instead of stopping processing on and suspending the record. Perceptive DataTransfer attempts to run the commands between *try* and *catch*. If it is successful, it omits the catch block. If it is not successful on any line of code, it moves to the next block and runs the commands in that block instead.

Example

```
try
   begin-select
      field 1 ~$field1
      from invalid table_name
    end-select
catch
   logit _GL.EXCEPTION_
end-try
```

In this example, Perceptive DataTransfer first attempts to run the begin-select command; the command is unsuccessful with an SQL exception because the table and column do not exist. When Perceptive DataTransfer reaches the exception, it moves to the *catch* block and instead runs the *logit* command.

After it runs the commands in the catch block, Perceptive DataTransfer continues to process the code that follows *end-try*.

Use this function with the _GL.EXCEPTION_ global variable, to which the exception message is assigned. It is populated only when it is inside a catch block. Refer to Understanding global variables for more information about global variables.