Getting Started

Getting started with Automation Anywhere (AA for short!) is easy. Using the topics in this section, you can create simple automated tasks that are useful and expandable to larger, more complex processes.

In This Section

- Understanding the Features and Benefits
- Introduction Automation Anywhere Client
- Verifying System Requirements
- Preparing and Installing the Automation Anywhere Client
- Communicating with the Server/Control Room
- Contacting Automation Anywhere

Understanding the Features and Benefits

Automation Anywhere is intelligent automation software, providing unparalleled performance and scalability for greater efficiencies in your business processes.

Capabilities: What Does Automation Anywhere Offer?

Automation Anywhere helps you to optimize the time spent on running tasks by providing the following capabilities:

- Transform a repetitive, 20-step process to a single mouse click or keystroke.
- Run scheduled tasks anytime by using powerful task scheduling, even when computer is locked.
- Automate scripts that exist in disparate formats.
- Easily manage automated processes across a single machine or multiple machines, and across a division or an enterprise.
- Create automation tasks in minutes: record keystrokes and mouse clicks, and use easy, point-and-click wizards.
- Easily manage automation users with a scalable server that provides true collaboration.
- Maintain centralized systems for backup, security, and resources.
- Capture and reuse automated processes in a central repository.
- Manage job scheduling and execute batch data processing with ease.
- Automate FTP/SFTP processes.

Unique Features: How Do We Do It?

Automation Anywhere provides the following features, making it the only solution in the market to offer complete automation capabilities:

- SMART Automation Technology that quickly automates complex tasks, with no programming required, and is highly reliable and accurate.
- Centralized Enterprise Control Room that makes it easy to set up and maintain automated processes, users, and client machines.
- Industry-leading Web Recorder and Object Recorder that recognize objects, even when they move position.
- Over 550+ actions available, including:

1. Active Directory support for user/client management
2. Conditions and loops
3. User prompts
4. File, database, and system management
5. High-speed replay of automated tasks
6. Automatic email notifications
7. Task chaining
8. Hotkey setup and management
9. User-defined and system variables
10. Advanced error logging
11. PDF automation

Benefits: Why Do I Want To Use This Solution?

- Enable your teams to be 30-80% more efficient.
- Eliminate human error in data entry processes.
- Improve collaboration between employees and divisions.
- Increase transaction speeds.
- Reduce costs and achieve higher ROI on your existing IT systems
- Integrate multiple, disparate systems and data, including legacy systems, ERP systems, partner data, and Web portals.

Value: What Can I Do With Automation Anywhere Client?

Automation Anywhere helps companies achieve process consistency across a team, department, division, or enterprise by automating and managing workflows, processes, and tasks.

Some of the processes you might consider automating include:

- Time-consuming data entry processes like:
1. Checking inventory
2. Re-ordering supplies
3. Formatting and running reports
- Logging into websites, searching websites, submitting web forms, or updating records on a website.
- Managing ERP front-end processes, including SAP, Oracle Applications, Peoplesoft, MS Dynamics, Salesforce, and more.
- Automating data transfers, including importing/exporting data between applications or files.
- Use the Automation Anywhere Object Recorder to manage and execute Windows controls as a background process.
- Automate routine backups and file management tasks.

Automation Anywhere is the most complete business process automation solution available today!

Also See...

- Installing the Automation Anywhere Client
- Installing Your License Key
- Registering a Client with the Server/Control Room
- Starting the Automation Anywhere Client
- Upgrading from the Trial Version
- Verifying Automation Anywhere System Requirements
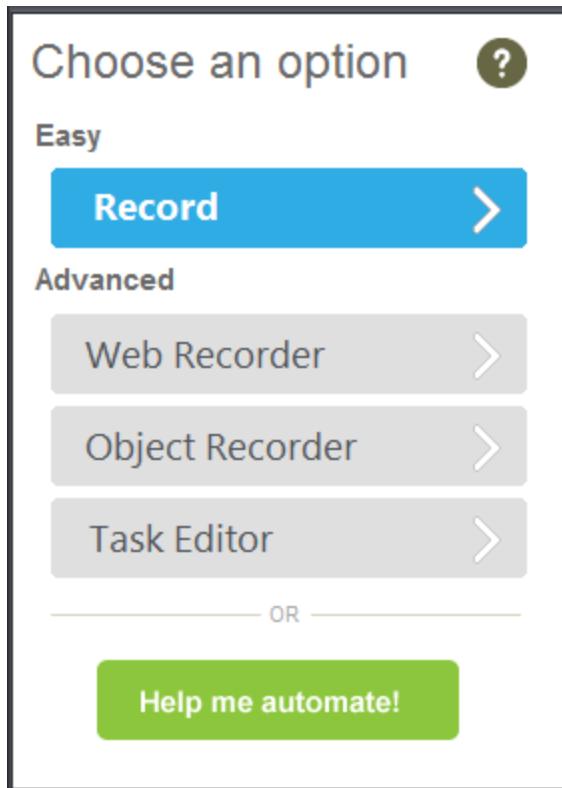- Creating an Automation Task
- Contacting Automation Anywhere

Introduction to Automation Anywhere

Automation Anywhere allows you to create automated tasks with ease. It's intuitive interface provides you the essential features to record, modify and run tasks that range from simple to more complex ones.

This topic will give you a bird's eye view of Automation Anywhere's 'Client Dashboard'. It also provides the necessary information to direct you to related topics.

Client Dashboard - A Bird's Eye View

As soon as you launch Automation Anywhere, it displays the Automate page:

(Image as seen in Enterprise Edition)

📝 Note: You can directly start creating tasks or open recent one's if any.

Upon pressing close, the Client Dashboard is displayed:

(Image as seen in Enterprise edition version 8.0)

Note: The user interface will vary depending upon the features available with the product. To compare versions, check out our **website**

The Client Dashboard basically comprises of the Menu Bar, Tool Bar, Folder Panel, Task Panel and various Feature Panels. Collectively these allow you to create, record, run and modify your tasks.

1. The Tool Bar (located below the menu bar) is a combination of tool buttons and functions for recording, running and modifying tasks:

a. Record tasks with different recording methods available on the 'Automate' window.

(Image as seen from version 7.5 onward)

Refer Selecting Recording Methods for details.

b. Record keystrokes as well as mouse clicks and movements.

Refer Using the Standard Recorder for details.

- Run the selected task

Refer Running a Task for details.

- Modify the selected task using the Task Editor.

Refer Using the Task Editor for details.

2. Access the most frequently used functions on the tool bar, such as:

a. **Web Recorder**

It understands all web controls thus enabling you to record web-only tasks.

Refer Using the Web Recorder for details.

b. **Object Recorder**

It understands all objects and windows controls. Record, manage, run Windows actions in the background and run other programs simultaneously.

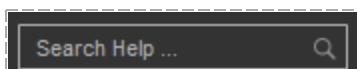Refer Using the Object Recorder for details.

c. **ROI Calculator**

It displays the return on investment (ROI) accrued through use of Automation Anywhere <text>.

Refer Using the ROI Calculator for details.

> Note: While recording and running automated tasks ensure that you follow best practices doc-
> umented in the help topic Best Practices for Recording/Running Tasks

Additionally, Automation Anywhere offers features to troubleshoot in the form of 'Search Help', 'Chat' and 'Client Notifications'. *

- Access the Automation Anywhere online knowledge base with the 'Search Help' feature.

Search Help ...

- Chat with our online Automation Experts with the 'Live Assist' feature.
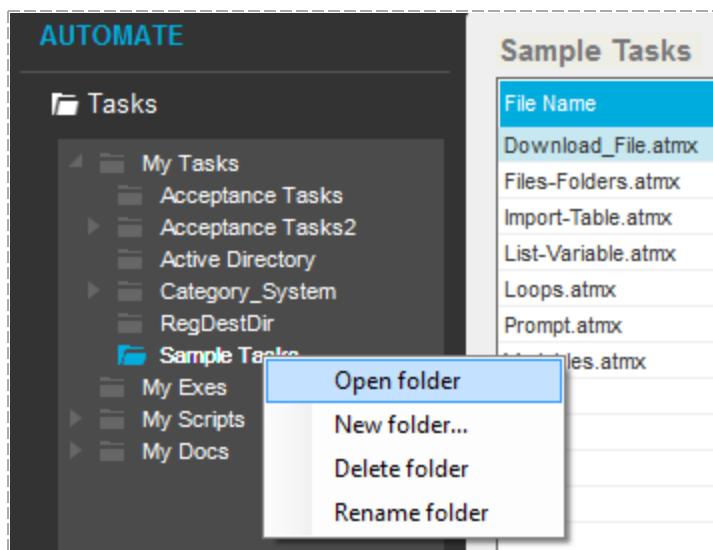
- Receive trial status, task failure and external notifications. Reference Automation Anywhere Client Notifications for details.

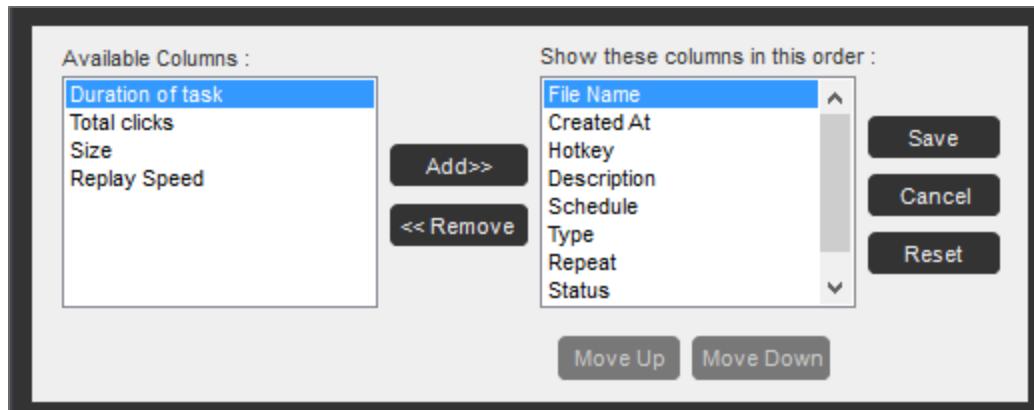*Available from version 7.5 onward.*

The Folders Panel

At the top left is the Folders panel, which lists existing tasks in the Task List panel on its the right. Right click a folder to open, cre-
ate, rename or delete that folder:

The Task List Panel

The Task List, which takes up most of the workspace, displays relevant task details in columns such as File Name, Description, Created At, Duration, Size, Hotkeys (if any) and Status etc.

💡 Tip: Select column view from View > Column Displayed

Located at the top-most right corner of the Task List are buttons that enable you to delete, create an exe and upload the selected task.

Alternately click the Action drop-down button to carry out task relevant actions such as run, edit, upload*, copy, rename, locate and send.

The Features Panel

Towards the bottom left are the product specific features:

a. **Repository***

   Communicate with your Server/Control Room to register, upload and download tasks, request access privileges etc.

   Refer Communicating with the Server for details.

b. **Reports #**

   It provides graphical reports that display the status of tasks and workflows, as well as the ROI over time.

   Refer Using the Report Designer for details.

c. **Trigger Manager/Triggers #**

   It provides a unified view of the triggers that are connected to your tasks. Use it to enable or disable selected or all triggers as well as add, modify and delete specific triggers connected to a task.

   Refer Using the Trigger Manager and Adding Triggers to a Task for details.

d. **Schedules #**

   It is a convenient facility to manage automated tasks on time based scheduling capabilities. You can schedule to run the task daily, weekly, monthly or with a trigger event.

   Refer Using the Schedule Manager for details.

e. **Workflows #**

   It helps you build process flow diagrams in a graphical environment. These you can subsequently use to build and run automated tasks in the program.

   Refer Using the Workflow Designer for details.

* Available with Small Business and Enterprise versions only.

 # Feature also available in Standard edition version 8.1.2 or higher.

Panel for Task Relevant Activities

Below the Task Lists is a panel that displays task related Properties, Schedule and Trigger in the form of tabs.

a. **Properties**

View and set general properties such as allocating priority status or specifics such as frequency/conditions to repeat the selected task, adjusting speed, email notifications, hotkeys, security parameters to ensure authorized users run the task etc.

Refer Setting Task Properties for details.

b. **Schedule**

Schedule your automated tasks to run at preferred times, without requiring supervision or intervention.

Refer Scheduling Tasks using the Scheduler for details.

c. **Trigger**

Connect a trigger type to the selected task. It automatically runs the task in response to an event that occurs on your computer.

Refer Adding Triggers to a Task for details.

An additional tab towards the bottom right is the Description/Notes panel, which allows you to enter free text for selected task. The text can be edited if and when you modify the task.

Refer Viewing and Setting General Properties for details.

Also See...

- Verifying System Requirements
- Checking the Current Software Version
- Upgrading to a Licensed Version
- Creating an Automation Task
- Using Special Keys
- Defining Client Hotkeys

Disclaimer: *The images displayed may vary from the actual product because of regular updates.*

Preparing and Installing the Automation Anywhere Client

These topics will help you determine if your system meets the requirements for installing the Automation Anywhere Client and will help you complete the installation.

Also See...

- Verifying System Requirements
- Installing the Automation Anywhere Client
- Checking the Current Software Version
- Installing a License Key
- License Agreement
- Privacy and Security

Verifying Automation Anywhere System Requirements

This section helps you determine whether your system has the proper hardware and software to install Automation Anywhere. Before installing the Automation Anywhere, verify that your environment supports the following requirements.
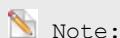
Operating Systems:

- Microsoft Windows Server 2012 (valid for AA Small Business and AA Enterprise)
- Microsoft Windows Server 2008 R2 (valid for AA Small Business and AA Enterprise)
- Microsoft Windows Server 2003 (valid for AA Small Business and AA Enterprise)
- Microsoft Windows 8.1/ 8 (Home and Pro Versions for AA Standard)
- Microsoft Windows 7 (Home and Pro Versions for AA Standard)
- Microsoft Windows Vista
- Microsoft Windows XP

   (Both 32-bit and 64-bit OS versions are supported)

Processor speed: Recommended - 3 GHz or higher.

RAM: Recommended - 4 GB or higher.

Hard Disk capacity: 200 MB of free hard disk space for installation.

> 📝 Note:
> - On an average an Automation Anywhere script is approximately 100-150 KB. Additional free disk space is required to develop automation projects, as Automation Anywhere creates temporary files like screen-shots, server logs, audit files etc. during the execution of the automation scripts.
> - The actual free space required increases with the project size and hence it is recommended to have at least 40-50 GB of free disk space in order to implement long term projects.

Additional Requirements

- 1024 x 768 or higher resolution monitor
- Mouse or other pointing device
- Microsoft Internet Explorer 8.0 or later.

.NET Framework Requirements

A .NET Framework is required to run Automation Anywhere products.

- .NET Framework 4.0
- .NET Framework 3.5 + SP1

> 📝 Note: If using Enterprise, you need to have **both versions** of .NET Framework installed; for Small Business, Premier and Standard (version 8.2 or higher), only .NET Framework 4.0 should suffice.

Synchronization between Client and Server

- To synchronize the time between the Automation Anywhere Client and the Enterprise Control Room, enable the Network Time Protocol (NTP) settings on your network.

   For more information modifying the NTP settings for your network, see your system administrator.

Also See...

- Installing the Automation Anywhere Client
- Checking the Current Software Version
- Registering a Client with the Server

- License Agreement
- Privacy and Security

Installing the Automation Anywhere Client

This topic describes how to install and uninstall the Automation Anywhere Client.

Installing the Automation Anywhere Client

To install the software, follow these steps:

1. Purchase and download the software from the Buy Now page.
2. Locate and double-click on the setup.exe file.
3. Follow the instructions in the installation wizard.
4. After the installation is complete, launch AA Client.

If you are currently using the trial version and you decide to upgrade to the licensed version of Automation Anywhere, you do not need to re-install the software. All functions and features will be unlocked after entering your license key.

Uninstalling the Automation Anywhere Client

To uninstall the Automation Anywhere Client, follow these steps:

1. On your Windows desktop, go to Start -> Programs.
2. Select your Automation Anywhere Product (the default program folder) and click on "Uninstall Automation Anywhere <Product>".
3. Follow the instructions in the uninstall wizard.

When you uninstall Automation Anywhere, all tasks that you have created are kept and maintained in the Automation Anywhere folder. If you no longer plan to use Automation Anywhere, you can remove these tasks by deleting the Automation Anywhere folder from the application path.

UAC (User Access Control) and DEP (Data Execution Prevention)

Windows introduced DEP (Data Execution Prevention) with XP SP2 and UAC (User Account Control) with Windows Vista. If the restrictions in these modules are set too high, it can restrict the overall functionality of Automation Anywhere.

```
 Note: These changes affect the security of your operating system environment. We recommend to
consult your local IT resource for more information about these functions.
```

The changes mentioned below will require you to reboot your computer. We therefore recommend, changing both settings at the same time in order to avoid multiple reboots.

To modify UAC settings:

1. Login as an Administrator.
2. Go to the Control Panel and select User Accounts.
3. Click on the option - 'Change User Account Control Settings'.
4. Drag the slider all the way to the bottom where it says 'Never Notify'.
5. Login as an Administrator.
6. Go to the Control Panel and select User Accounts.
7. Click on the option - 'Change User Account Control Settings'.
8. Drag the slider all the way to the bottom where it says 'Never Notify'.

   For more on the same, refer Setting User Access Control

To modify DEP settings:

1. Login as an Administrator.
2. Go to the Control Panel and select System.
3. Click on the option - Advanced System Settings.

4. Under the 'Advanced' tab, click on the 'Settings' below the 'Performance' area.

5. Go to the 'Data Execution Prevention' tab.

6. Select the radio button - 'Turn on DEP for all programs and services except those I select:'

7. Click on the 'Add' button and add the Automation Anywhere.exe, AA Player.exe and AA Task Editor.exe from the 'Program Files (x86)\Automation Anywhere Enterprise7.1\<Product>' folder to the allowed list.

8. Click on 'Apply' and then 'OK'.

9. After the reboot, right click on the Automation Anywhere Client icon on your desktop or under the 'Start' menu and select the 'Run as Administrator' option to launch the program.

Also See...

- Registering a Client with the Server
- Verifying System Requirements
- Checking the Current Software Version
- Installing a License Key
- License Agreement
- Privacy and Security

Upgrading to the Licensed Version

The Trial version of Automation Anywhere is available for download free-of-charge. This version contains most of the features that are available in a regularly licensed version. However, several features are locked and/or offer only limited use.

Benefits of Upgrading

The following capabilities are activated when you upgrade:

- Add password protection to tasks, which limit those users who can edit or run them
- Convert tasks to EXE files (standalone executable tasks)
- Access product support resources
- Certain Loop command functionality
- In addition, registering your license enables you to:

1. Take advantage of free or discounted software upgrades.
2. Receive announcements of new product features and upgrade offers.

Upgrading to Licensed Version

To purchase the Automation Anywhere Client, visit the Buy Now page.

After completing the registration, you will receive an email containing a registration license key/file.

To run the full version of Automation Anywhere, see Installing Your License Keys.

Also See...

- Installing Your License Keys
- Verifying System Requirements
- Installing the Automation Anywhere Client
- Checking the Current Software Version
- Registering a Client with the Server
- License Agreement
- Privacy and Security

Installing Your License Key

This topic describes the two methods recommended to install a license file, in order to register and activate the full version of Automation Anywhere Client.

After the completion of your order, you will receive an email from Automation Anywhere containing a registration license file. Download and save the file to a desired location on your computer.

You can install the license file using one of the following two methods:

After starting Automation Anywhere Client for the first time

1. Launch Automation Anywhere Client from the icon on your Desktop **or** from Start -> All Programs -> Automation Anywhere Enterprise 7.1 -> AA Enterprise Client.

2. On the initial screen, Click on the 'Install License' button.

3. Click on the 'Browse' button and select the license file.

4. Click OK.

Using the Help Menu

1. Under Automation Anywhere Client, go to the 'Help' menu and select 'Install License'.

2. Click on the 'Browse' button and select the license file.

3. Click OK.

Features and capabilities that were locked in the Trial version are now available for use.

> 📝 `Note: While installing the license file, if you receive a security pop-up notification advising that the UAC (User Access Control) is turned ON, please visit` *UAC* `and` *DEP* `sections in the knowledge base.`

Also See...

- Upgrading from the Trial Version
- Verifying System Requirements
- Installing the Automation Anywhere Client
- Checking the Current Software Version
- License Agreement
- Privacy and Security

Login to Automation Anywhere Client

Available in Small Business and Enterprise editions 7.5 onward.

Automation Anywhere creates an environment wherein it auto-detects your server connections to enable you to register a new client user and login.

Your client thus seeks out the available Control Room/Server and logs into it. You only have to enter your credentials to register and login.

> Note:
> - Registration is automatic for Manual and Active Directory users on login.
> - However, a domain name change post registration will require re-registration. **Learn More**

Creating a Client User in Control Room/Server

1. Create client user in Client Control Center/ Client Manager.

   Refer Adding a Client for details on how.
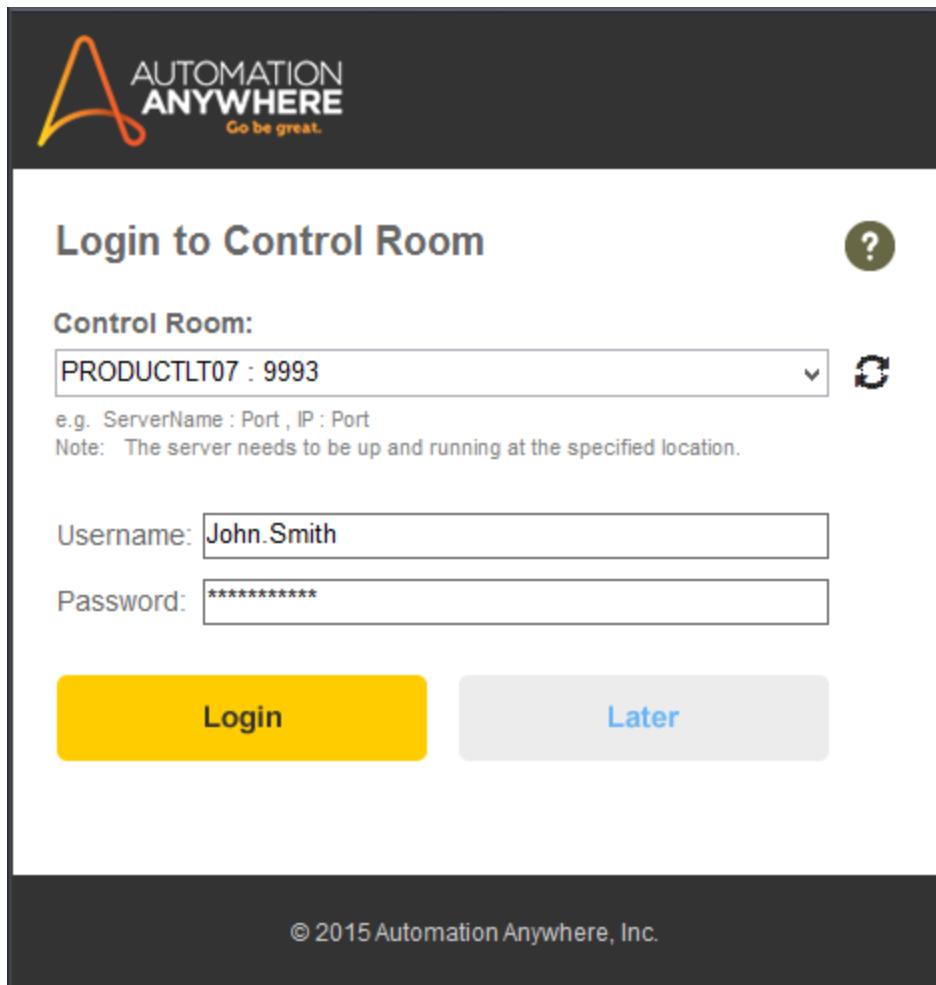
2. Launch Enterprise/Small Business Client.

3. Your Client auto-detects available server connections.

   If launching for first time, select the most appropriate one from the list that is populated on auto-detection. On logging in the second time, your last login Server/IP is displayed.

   If you wish to login using another server connection, click 'Refresh' to auto detect server connections. Alternately, input the required values.

> Note: If your client and server reside on the same machine you can input 'localhost:<Port>'.

(Image as seen in Enterprise edition 8.1

- If already logged in, as different user, click Tools > Options > Server Settings. Click Re-login and enter credentials.

(Image as seen in Enterprise edition 8.0)

Note: The logged in username and status is displayed in the task bar.

(Image as seen in Enterprise edition 8.0)

Verifying Client Registration in Control Room/Server

1. Enterprise Control Room

You can verify whether your client has registered successfully or not in the Client Information section of Client Control Center.

Note: Click 'Refresh' to update.

2. Small Business Server

You can verify whether your client has registered successfully or not by launching the Client Manager on the Server:
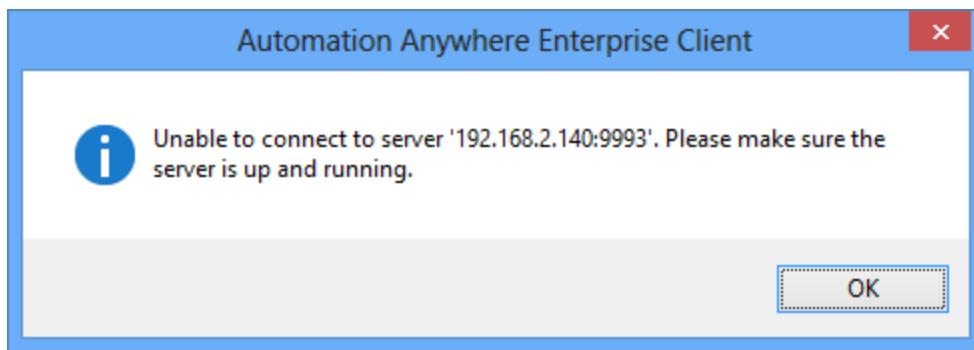
Unable to Login?

If, during login, the client displays the following message, you will have to verify whether your Client and Control Room/Server are configured to the same version:
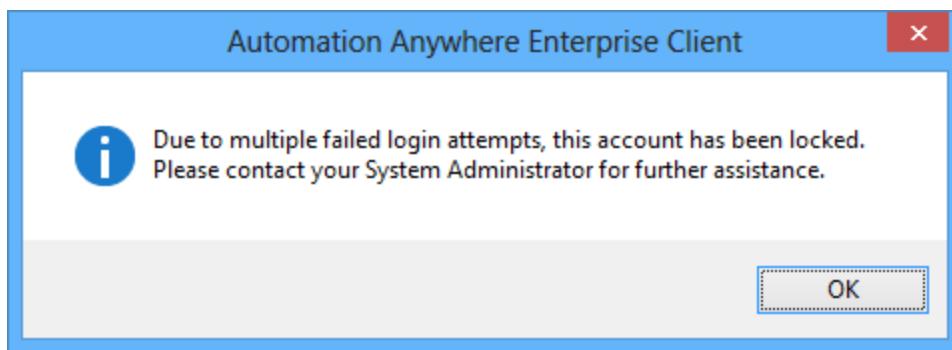
If the versions mismatch i.e. your client is of the latest/higher version, you will have to upgrade the server to the same version. For details, contact support@AutomationAnywhere.com

NEW! Policy for multiple login attempts by a Client*

* Available in Enterprise edition 8.1 onward

To maximize the security of a Client Account, a Client user can be allowed only a specific number of attempts to login. The number of invalid attempts are defined by the Control Room administrator in the Control Room.

If your Administrator has enabled this policy, your account gets locked once you exceed the allowed number of unsuccessful logon attempts. At this point, the following message will appear:

In order to unlock the account, you are required to inform the Control Room/System Administrator. Your status in the Client will, meanwhile be 'Offline'.

> Note: Account Lockout can be defined for both - Manual user and Active Directory users.

Refer Using Client Control Center for details.

Also See...

- Adding a Client
- Using Client Control Center
- Using Server Settings

Using the Automation Anywhere Account Login

Available from version 7.5 onward

Creating your Automation Anywhere Account ensures that you are connected to your Client, online Automation Anywhere Account and Support site.To benefit from the collaborative features of the product, login via a 'Single Sign On' - your 'Automation Anywhere Account' .

Manage various activities such as downloading updates, upgrading to a higher edition, changing the settings or accessing the support site to log in your ticket or comments.

Signing up for an account

A.   For Enterprise and Small Business Editions
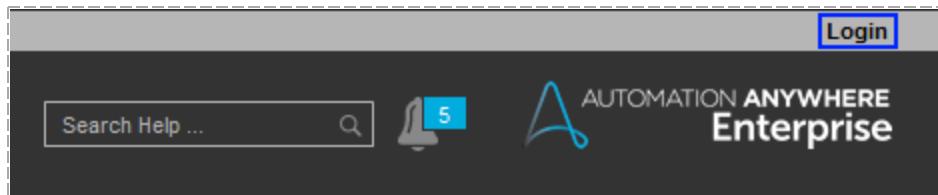
1.   Register and log in into your Client, if you are an Enterprise or Small Business user. Reference Register and Login to Automation Anywhere Client for details.
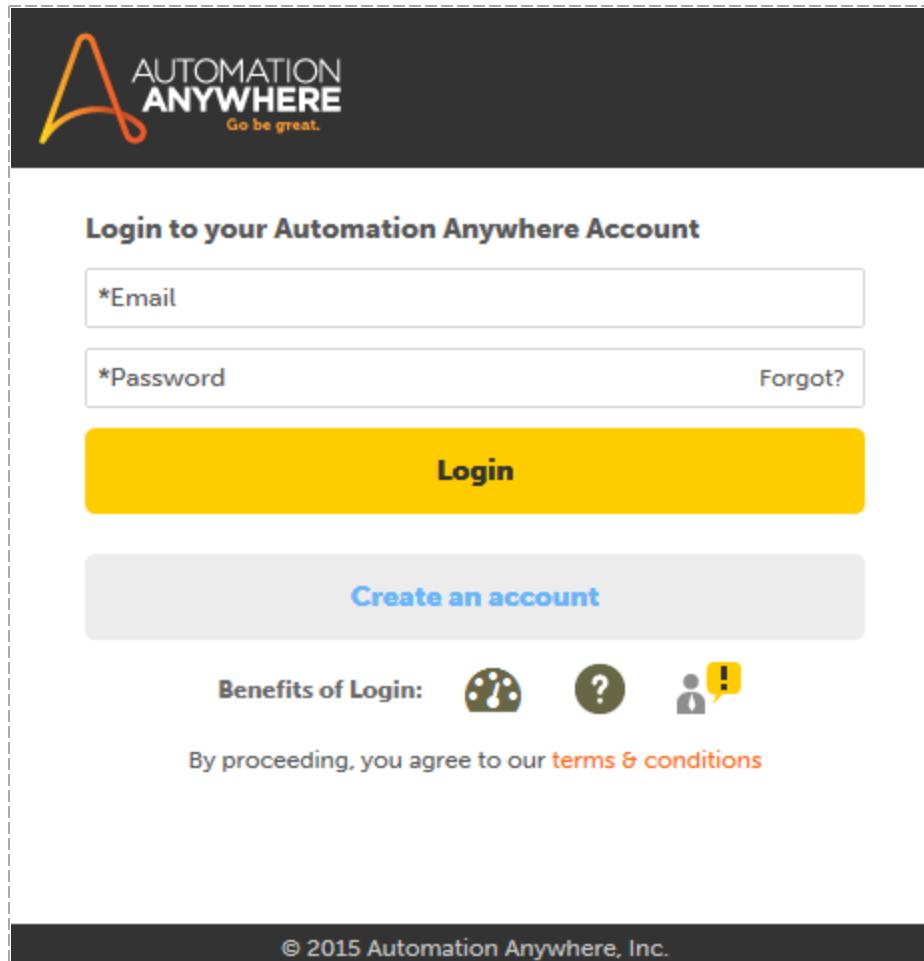
> 📝   Note: In Standard edition*, the 'Login' window is invoked as soon as you launch the Client. Also, you must 'Login' to be able to use the application.
>
> *Earlier available in Premier (up to version 8.1.2)*

2.   To create your online account, click 'Login' in the topmost right corner of your Client.

3. Click the 'Create an account' link at the bottom in the 'Login' window.

4. Automation Anywhere will guide you to the 'Sign Up' page. Here, you are required to provide your credentials. Click 'Submit' once done.

Sign up

*First Name

*Last Name

Already have an account? Login

Company

*Email

*Country

*Phone

Submit

Contact   End User License Agreement   Uninstall Instructions

5.   To activate your account, a link will be sent to the email address that you provide. Click 'Resend' if the email address is incorrect.

We've sent you the account activation link

Check your email for a link to activate your Automation Anywhere account.

In case the below email is incorrect, please modify and click Resend Link.

AA001@automationanywhere.com

Resend Link

6.   After activating your account by clicking on the link in the mail you receive, you will be guided to the 'Finish creating your account' page.Set your password and click on 'Submit' to conclude the sign up process.

Finish creating your account

To activate your account and download your free trial, simply choose a password and then hit the button below.

AA001@automationanywhere.com

*Password

Password to contain:
1. Length: 4-15 characters
2. Allowed Characters: (a-z,A-Z,0-9,@,-,_and.)
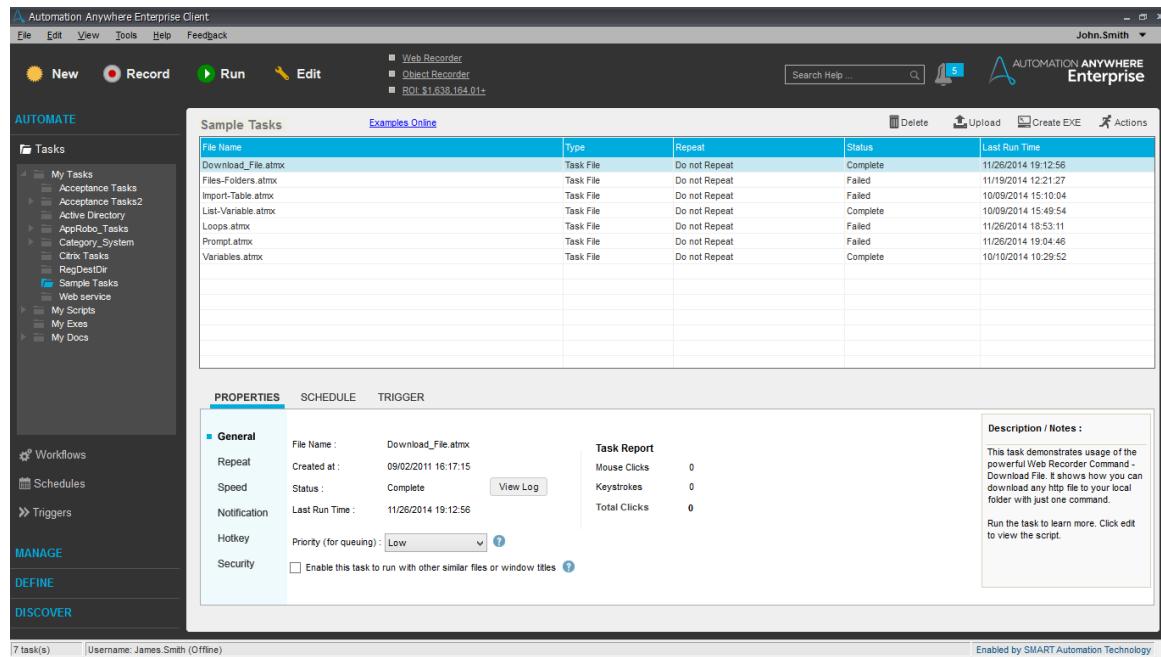
Submit

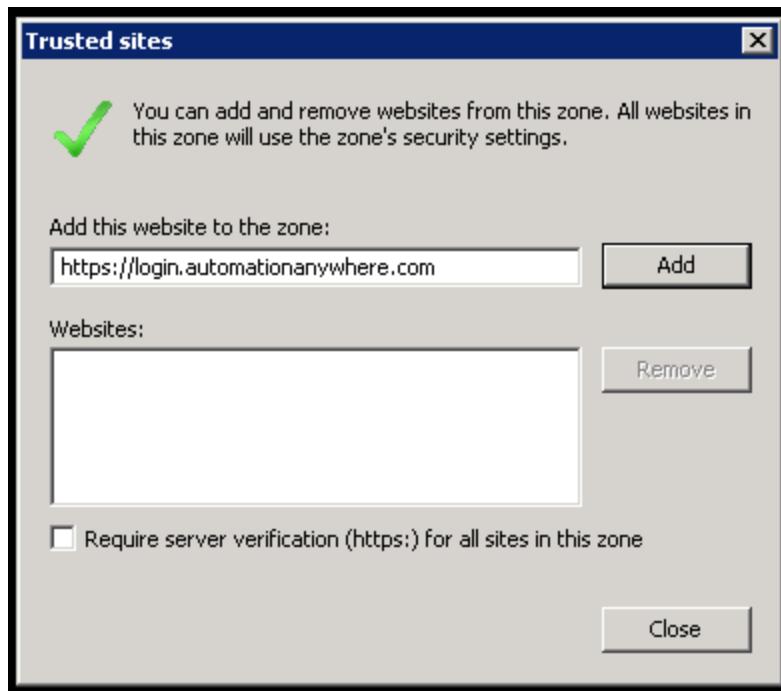7.   Launch the Client and provide your credentials to log in.

B. **For Premier and Standard Users**

In Standard or Premier, the 'Login' window is invoked as soon as you launch the Client. Also, you must 'Login' to be able to use the application.

To create an account follow steps 3 through 7.

C. **For Windows Server 2008 R2 Enterprise**

If you are on Windows Server 2008 R2 Enterprise and if the 'Internet Explorer Enhanced Security' is enabled, you will be required to configure Automation Anywhere website in the trusted sites list in order to be able to login.

Reference how to enable/disable enhanced security here.

Managing your Automation Anywhere Account

After activating your account, you will be guided to the 'My Products' section of your account, wherein you can manage all your account related activities such as downloading updates, upgrading to a higher edition, changing the settings or going to the support site to log in your ticket or comments.

Resetting Account Password (from Client)

If, for any reason, you are unable to log in to your account or have forgotten your password, you can reset it by clicking the 'Reset Password' link in the 'Login' window.

1.   Enter your email credentials to send a link for resetting the password.

2. Click on the link given in the mail.

3. Enter new password details.
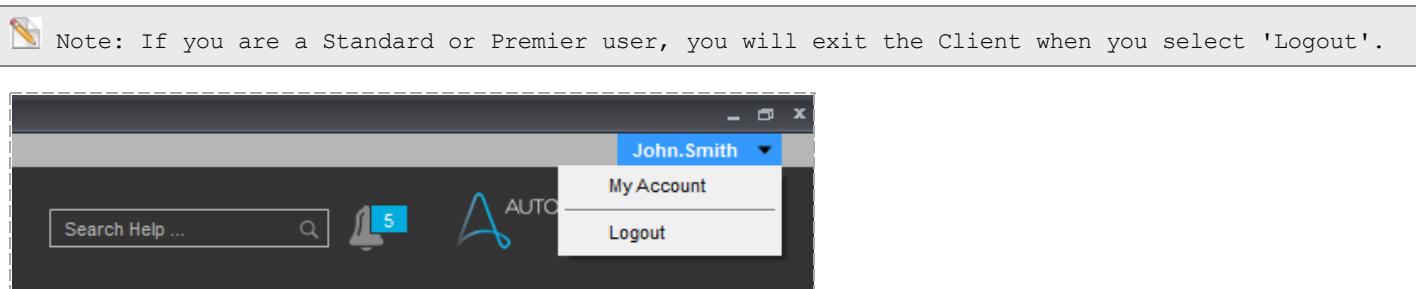
> Note: Refer the password setting criteria as indicated.

Automation Anywhere Account - A Client Perspective

Once you are registered and logged into your client, you can manage your account from within the client. The login name displayed on the top right corner of the client is an indicator that you are logged in.
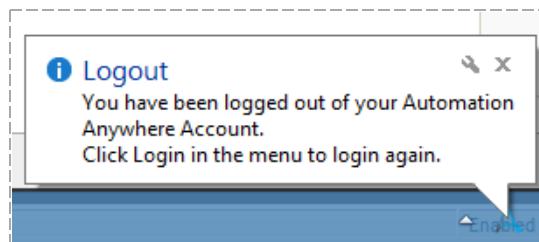

If you click on the drop down, you will be able to either go to your account or logout.

Note: If you are a Standard or Premier user, you will exit the Client when you select 'Logout'.

Note: If you are not registered or are not logged in, the status will be 'Login' in the right topmost panel.

You know you are logged out when the following message is displayed in the task-bar of the client to the bottom right:

Also See...

- Introduction to Automation Anywhere
- Installing the Automation Anywhere Client
- Register and Login to Automation Anywhere

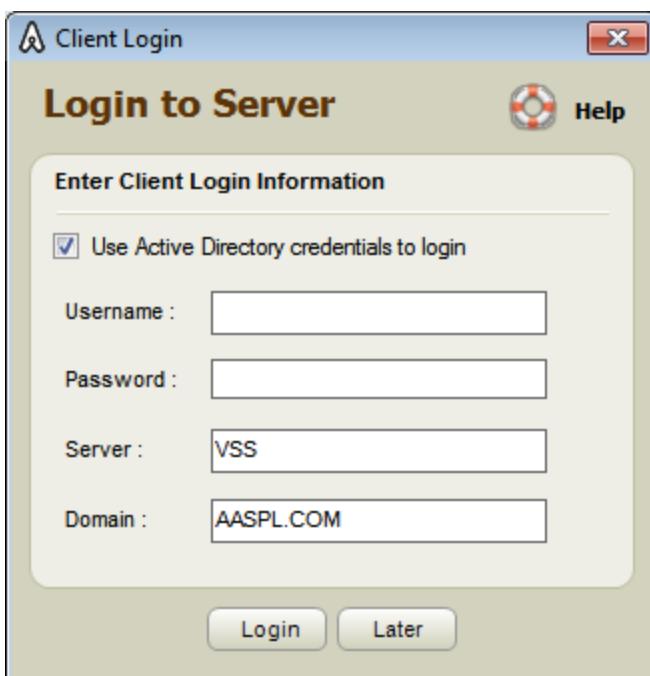# Starting the Automation Anywhere Client

Applicable to Small Business and Enterprise editions up to version 7.1

This topic describes how to launch the Automation Anywhere Client.

On installation the program is displayed as an icon on your desktop.

You can begin using the program by following these steps:

1. Launch the Automation Anywhere Client program using any one of these actions:

- Locate the Automation Anywhere Client icon on the desktop and double-click on it.

   or

- Click on the Windows Start button in the lower left corner of your desktop, locate the Automation Anywhere Client program, and click on it.

2. Now, the 'Client Login' is displayed. You can login by entering your Username and Password. You can also login by selecting the 'Use Active Directory Credentials' check box.

> Note: You have the option of logging onto the server later by clicking 'Later'.

3. An initial screen is displayed with options to begin creating tasks. You can close this window and begin using the main client screen, or begin working from this screen.

> Note: If you do not want the initial screen to be displayed in the future, select "Don't show this 'Start Page again" check box.

Your Automation Anywhere Client is now up and running.

Also See...

- Recording, Editing and Running Tasks
- Using Special Keys
- Exiting the Automation Anywhere Client

Recording, Editing and Running Tasks

This topic describes how to record and run basic automation tasks.

Recording and running simple automation tasks involves three basic steps:

1. Record
2. Save
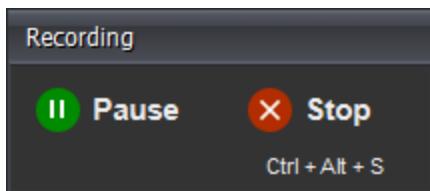3. Run

Recording an Automated Task

To record an automated task, follow these steps:

1. For recording simple tasks and which can be run on the computer they are recorded on, click the record button at the top of the main client screen.
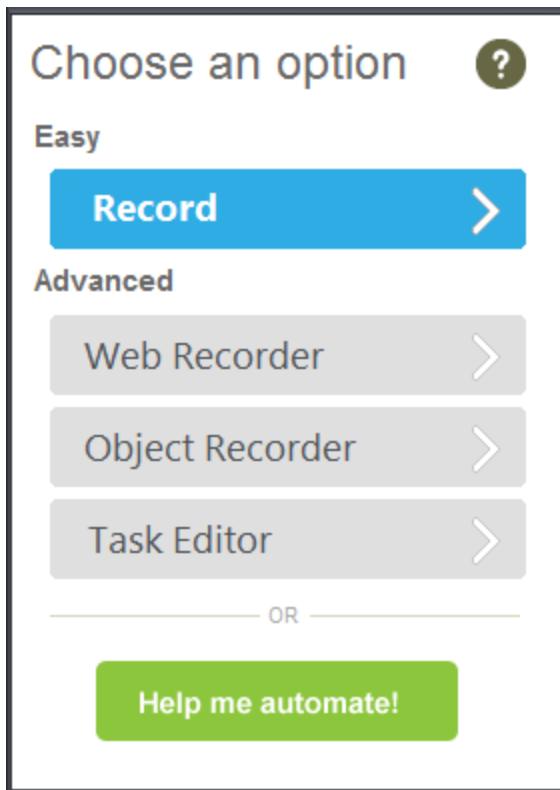

   Refer Using the Standard Recorder for details.

- Automation Anywhere begins recording your activity, including keystrokes, mouse movements and mouse clicks.
- During this process, the Automation Anywhere client window disappears from view. A small "Stop Recording" window appears in the bottom right corner of your desktop.

Note: You can also create a new automated task by clicking the New Task button at the top of the client screen.

- A window is displayed that enables you to select from several methods for creating a task:

Choose an option ?

Easy

Record 〉

Advanced

Web Recorder 〉

Object Recorder 〉

Task Editor 〉

———— OR ————
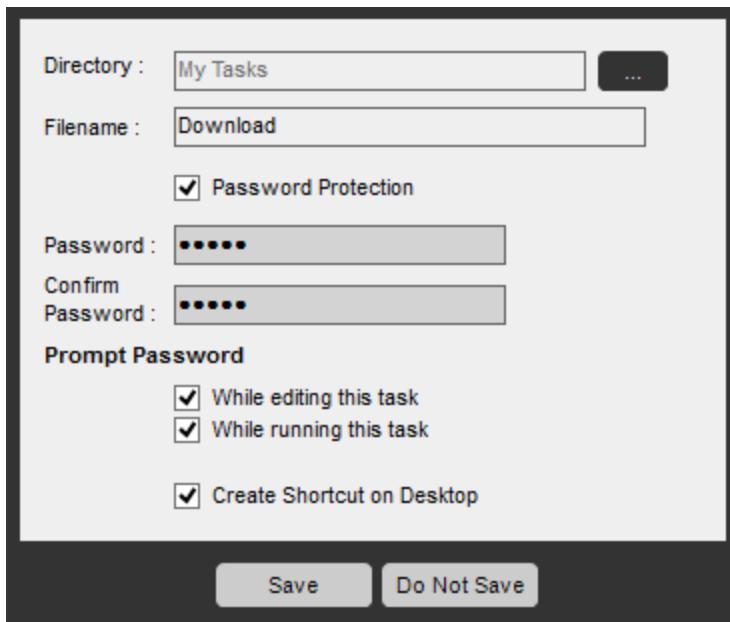
Help me automate!

*(*Image as seen from version 7.5 onward)

For more information on choosing a recording method, refer Selecting a Recording Method. Alternately, you can click the 'Help me choose an option' link to go to 'Live Assist' - our support chat option.

2. Perform the actions that you want recorded for automation. You can open applications, open new windows, click on buttons, fill in forms, search a website, and perform any other actions you want.

3. Click the stop button to stop recording the task.

× Stop

- A Save Task window is displayed.

4. In the Save Task window, type the name of the directory where you want to save the task, or accept the default directory (My Tasks).

5. Type a file name for your task and click on Save.

6. If necessary, you can enter a password to protect the task.

- Your task is now created.

Creating a Shortcut for a Task on Desktop

It is possible for you to provide a shortcut on your desktop to run a saved task. An icon is associated with the task to launch it quickly.

To create a shortcut on the desktop, follow these steps:

1. In the Automation Anywhere Client window, select your task in the Task List.
2. Right-click on the task, and choose "Send To -> Desktop (Create Shortcut)".
3. Locate the icon on the desktop and drag it to the desired location.

Running Your Recorded Tasks

You can run the task that you recorded by choosing one of the following methods:

- Select the task in the Task List and click Run.

Or

- Double-click on the task in the Task List.

Or

- Double-click on the desktop icon.

Stopping a Running Task

To stop a task from running, take one of the following actions:

1. Press and hold the ESC (escape) key for 2 to 3 seconds.
2. In the run-time status window, click on the Pause button.

Editing a Task

After you initially create an automated task, you might want to further customize or optimize the performance of your task. You can edit the task file without the need to re-record it.

- To make changes to your task, select the task in the Task List and click the Edit button.

- The Task Editor is launched, enabling you to add functions, remove delays, and add action sequences.

For more information, see Using the Task Editor.

Also See...

- Installing the Automation Anywhere Client
- Starting the Automation Anywhere Client
- Using Special Keys
- Adding Properties to a Task
- Scheduling a Task to Run
- Exiting the Automation Anywhere Client

Adding Properties to a Task

You can add properties to increase the efficiency of a task. This topic describes in brief the properties that can be added to a task. It also guides you to appropriate topics to enable you to directly access the one you wish to peruse.

Add the properties given below to help increase the performance, functionality and usability of your task:

1. **General:** View general properties such as file name, created at, last run time and log on the General tab. Also, change the priority* of the task to determine it's precedence for queuing.

   Available for Enterprise version only.

   Refer Viewing and Setting General Properties

2. **Repeat:** You may need to repeat a task for instance when extracting data from a website which is updated frequently. Specify when running a task it should repeat particular number of times. Also specify that the task should repeat the run any number of hours, minutes, or seconds.

   Refer Repeating a Task

3. **Speed:** Specify how quickly a task should run. At times certain tasks require precise mouse movements. While recording, your movements might be slow. After recording such a task you have the option of replaying the same at varying speed.

   Refer Adjusting the Speed of a Task

4. **Notification:** You can specify that you want Automation Anywhere to send an e-mail to a particular e-mail address after the task completes a run.

   Refer Sending Email Notifications for a Task

5. **Password Security:** You can make your task secure by requiring a user password to run and edit the task.

   Refer Setting Security Features for a Task

6. **Hotkey:** Assign a keyboard shortcut to launch your automated tasks with the press of a single key termed as 'hotkey' in the program.

   Refer Setting up Hotkeys for a Task

7. **Screen Security*:** You can specify that Automation Anywhere run the task without displaying the actions on the screen during the run. You can also disable the keyboard and mouse while the task runs.

   Available with purchased version of Enterprise Client only.

Also See...

- Running a Task
- Using Special Keys

Using Special Keys

While recording, it is possible to increase the productivity of a task, by using special keys for a task.

The following special keys are available under Automation Anywhere while recording and playing automated tasks:

Esc - To stop (abort) a running task, press and hold the ESC key for 2 to 3 seconds. Automation Anywhere also provides you with an option to configure your own hot key to stop an automation task from running.

The next time you run an aborted task, Automation Anywhere starts it from the beginning

Pause Break - To pause an automated task while it is running, press the Pause/Break key or click on the 'Pause' button on the Run Time window. Once you pause a task, a 'Resume' button appears on the Run Time window.

To continue running the task, click the Resume button.

```
Note: This feature is useful when you run long tasks that require interrupting in order to use
the machine for other purposes.
```

Also See…

- Installing the Automation Anywhere Client
- Starting the Automation Anywhere Client
- Recording, Editing and Running Tasks
- Exiting the Automation Anywhere Client

Enabling Remote Desktop Connections

This topic provides information about using Remote Desktop applications to run your automated processes.

Running Automated Tasks under a Minimized Session

When you minimize a remote desktop window (the window that displays the desktop of the remote computer), the operating system switches the remote session to the non-GUI mode and does not display windows and controls.

As a result, Automation Anywhere is unable to interact with the application GUI. At this point, the automated GUI task fails.

To run GUI tasks on a remote computer while minimizing the remote desktop window, change the registry keys on the computer from which you connect to the remote Automation Anywhere workstation. Complete the following steps:

1. Log into the computer from which you connect to remote computers.
2. Close all open remote desktop sessions.
3. Click Start -> Run (or Windows + R).
4. In the Run dialog window, type "regedit" and press Enter.
- The Registry Editor starts.
5. Navigate to one of the following registry keys, depending on whether you want to modify the remote desktop settings only for the current user or for all users on the computer:
- HKEY_CURRENT_USER\Software\Microsoft\Terminal Server Client
- HKEY_LOCAL_MACHINE\Software\Microsoft\Terminal Server Client
6. Create a DWORD value named RemoteDesktop_SuppressWhenMinimized and set it to 2.
7. Close the Registry Editor.
- When you minimize the Remote Desktop Connection window on your computer, the remote computer GUI should not be affected, making the GUI available to your automated tasks.

> Note: Changes made to the registry can affect the overall performance and function of the system. Automation Anywhere is not responsible for adverse system behavior that results from making registry changes. Working with your system administrator is advised.

Using VNC - An alternative to Remote Desktop

In general, Remote Desktop applications do not allow automation software to run or perform tasks when you minimize or close the window. Remote Desktop creates a session for every connection made to the remote machine. Under this session, it provides the connecting machine with complete rights to control the remote machine.

However, when disconnecting from or minimizing the remote window, Remote Desktop retrieves the rights and any access to the GUI of the remote machine is lost. This causes automated tasks to lose control of the remote system, and they do not run as expected.

> Tip: Use Virtual Network Computing (VNC) to connect to the server. VNC allows you to run automated tasks even when the session is minimized or disconnected.

- VNC establishes connections with the remote system in a different manner from Remote Desktop. VNC physically connects to the remote system, rather than creating sessions for each connection. The connection retains controls on the remote GUI, even when the VNC window is minimized or disconnected.
- Your automated tasks can continue to run in the same manner as they do on a local machine.
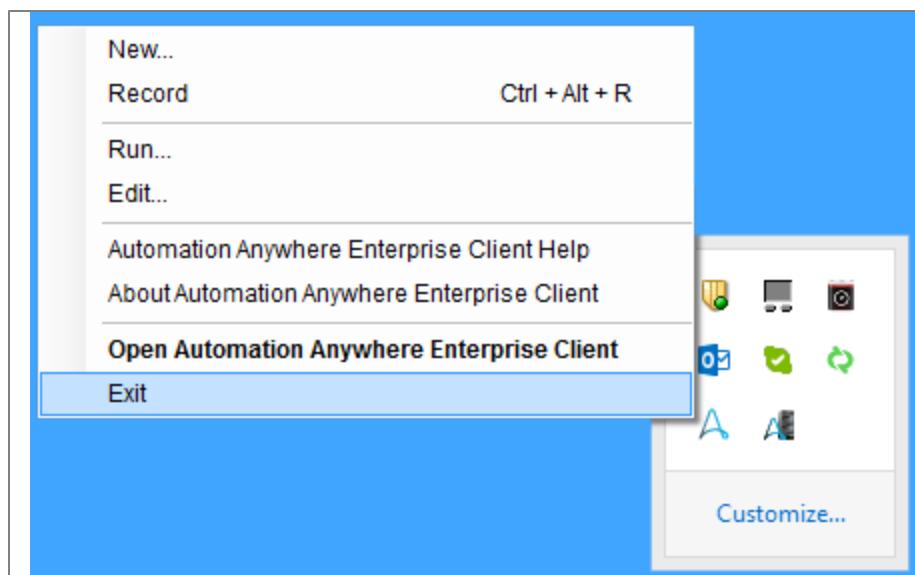- For more information on VNC, see: http://www.realvnc.com/download/vnc/

Exiting Automation Anywhere

This topic describes how to close and exit Automation Anywhere.

You can exit the program by taking one of the following actions:

1. Click the "X" in the upper right corner of the client window.
2. Click on the File menu, and select the Exit option.
3. Simultaneously press the ALT+F4 keys on your keyboard.

> Note: To exit out completely from Automation Anywhere, right click on the Automation Anywhere icon in the System Tray available on the bottom right corner of your screen and select the 'Exit' option.

Also See...

- Starting the Automation Anywhere Client
- Installing the Automation Anywhere Client
- Recording, Editing and Running Simple Tasks
- Using Special Keys

Contacting Automation Anywhere

Automation Anywhere wants to make sure that your questions are answered in the most timely fashion.

When you have a question or technical issue, you have the following options for getting help. Refer the following for details:

- Have a Question?
- Getting More Help

Give Us Feedback - We would love to hear from you!

More ways to contact us:

Sales: Toll Free (USA): 1-888-484-3535

International Customers: 1-408-834-7676

Fax Number for Purchase Order: 1-408-834-7671

Media and Press: Visit Press Room

Send all payments to:

Automation Anywhere, Inc.

P.O. Box 41363

San Jose, CA 95160

Headquarter and other offices:

Automation Anywhere, Inc.

2025 Gateway Place, Suite 252

San Jose, CA 95110

USA

Have a Question?

Do you have questions about Automation Anywhere?

- Using the software and its features
- Installing or configuring the software in your environment
- Creating your automated tasks
- Purchasing questions

Use any of the following methods to contact us:

- Use the Automation Anywhere Knowledge Base Help System
- Learn from Online Examples
- Contact Technical Support

Contacting Technical Support

Automation Anywhere is committed to outstanding customer care. Our support team wants to resolve your issues as quickly as possible.

Depending on the nature of your problem, please take these steps:

1. Check the Automation Anywhere Knowledge Base Help System, including Frequently Asked Questions
2. Learn from Online Examples for quick solutions to common automation use cases
3. View Online Demos and Videos
4. Open a Support Case

How to Open a Support Case

To open a support case, you may contact Technical Support. A support representative will contact you as soon as possible.

Getting More Help

For more complex questions, use one of the following Automation Anywhere services:

1. Request a Live 1-on-1 Demo (in most cases, no fees charged)
2. Use the Ask the Expert Service (in most cases, no fees charged)
3. Use Automation Anywhere Consulting Services (fee-based services)

Contact us for pricing questions for these options.

Requesting Live 1-on-1 Demos

- Do you have a specific question about an Automation Anywhere feature?
- Are you confused about how to automate a particular process?
- Do you have questions about how to work with applications while automating processes?

For help with any of these issues, contact the Automation Anywhere Support team.

Our team members are happy to demonstrate features, answer your questions, and show you various automation scenarios.

Request a Live Demo now.

Is your issue more complicated?

Learn about our Automation Anywhere Consulting Services, and how we can help you to automate more complex processes.

Using our Ask the Expert Service

- Do you have a simple question about an Automation Anywhere feature?
- Are you trying to automate a process that involves common applications, such as Word, Excel, or other Microsoft Office products?
- Do you have a question about your account, your software version, or your environment?

If you need help with any of these topics, contact the Automation Anywhere Ask the Expert team.

Our knowledgeable team members are eager to answer your questions and help you quickly automate your processes.

Is your issue more complicated?

Learn about our Automation Anywhere Consulting Services, and how we can help you to automate more complex processes.

Using Automation Anywhere Consulting Services

The Automation Anywhere team is led by industry experts in the business process automation field. We've helped major companies from around the world automate their key business and IT processes.

The Automation Anywhere team is focused on streamlining the implementation of automation, and maximizing customer ROI.

We bring years of experience and best practices in dramatically reducing implementation time to ensure your team is successful.

Our services consultants are ready to provide expert, efficient hands-on training and implementation.

Both on-site and remote implementation services are available that provide:

- Implementation and consulting for simple and complex automation projects
- An unwavering focus on your company's objectives
- Best practices for business and IT process automation that maximize ROI and minimize risk
- When you engage the Automation Anywhere Consulting team, you are assigned a dedicated customer agent who is familiar with your domain, requirements, technologies, and environment.
- If appropriate, your consultant can remotely log into your computer to help you automate processes and guide you with your project.

Automation Anywhere helps you to automate any of the following processes:

1. Business processes
2. IT processes
3. Extraction of data from the Web
4. Data transfer from one application to another
5. ERP (Enterprise Resource Planning) data entry and integration processes
6. File and data management processes

Find out more about our consulting services by submitting the Consulting Services Request Form. A representative will contact you shortly.

Anywhere Monitor

Available from version 8.0 onward

Monitor your automation status on any device anytime.

What is Anywhere Monitor?

Anywhere Monitor consists of a Task Status widget that provides a dashboard view of your most recent automation task statuses in your Automation Anywhere Account on the cloud. Additionally, you can also track these statuses over a period of time (last 30 days). The task summary displayed in this widget can be used for analysis and planning for course correction, if necessary.

The Task Status Widget summary consists of:

- Task Name
- Task Duration
- Date
- Task Status
- Error Data if the task failed

> Note: You can avail these benefits only when you login to the client using Automation Anywhere Account.

How does Anywhere Monitor display my task details?

The task status information is automatically and securely shared and displayed in your Automation Anywhere account.

If you are an Enterprise or Small Business user, your administrator can choose to enable/disable the Anywhere Monitor feature from the Control Room (if Enterprise) or Server (if Small Business).

To know how to enable the feature,

- Enterprise Client reference Managing Server Using Server Status Portal
- Small Business Client reference Managing Server Settings

> Note: To use Standard edition*, you have to log in using Automation Anywhere Account. Hence, task status information sharing happens automatically in those products and cannot be turned off.
>
> *Earlier available in Premier (up to version 8.1.2)
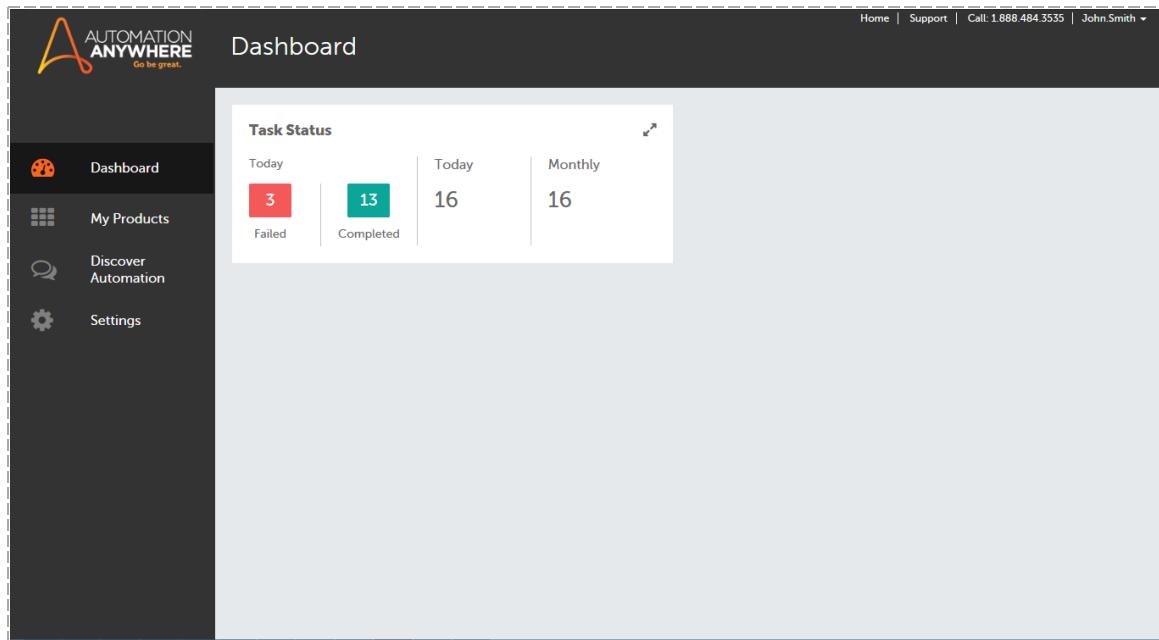
Anywhere Monitor Features

Dashboard - Task Status Widget

To view the Task Status widget, open your Automation Anywhere account by:

- Using a browser and logging in to my.automationanywhere.com
- Clicking on My Account link in the quick access menu on the top right (assuming you are already logged in to the application using Automation Anywhere account)
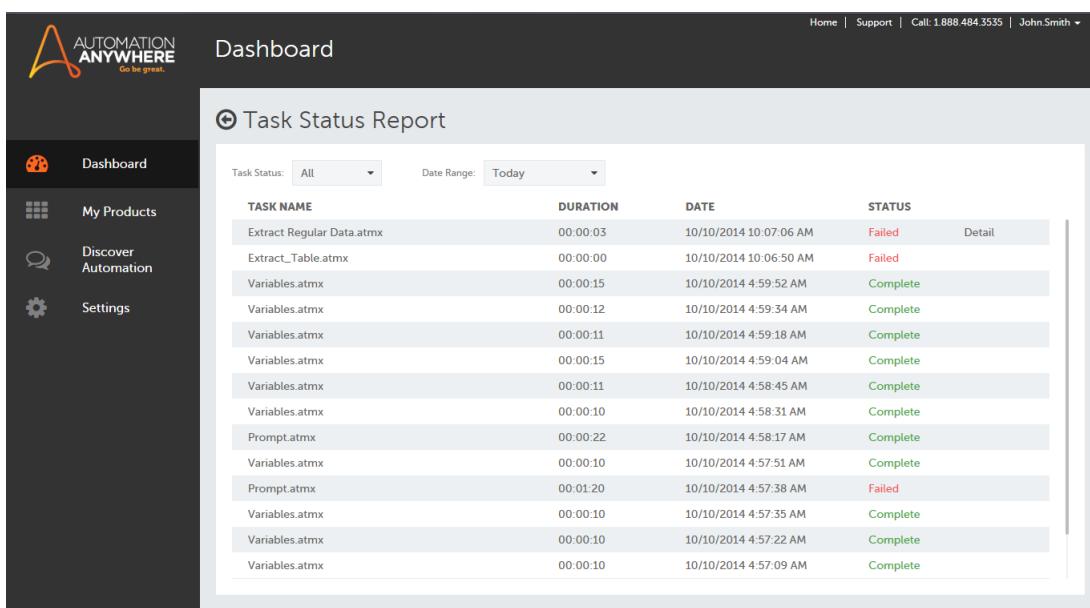
On the Dashboard, the Task Status widget is displayed by default as seen in the following screen shot(s):

By clicking on , the Task Status widget gets expanded to a Report widget and displays the following information for the selected date range and task status:

- Task Name
- Task Duration
- Date
- Task Status
- Error Data if the task failed
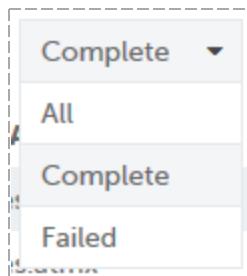
On the Task Status Report widget, you can set the following filters to get more relevant information:

- 'Task Status' filter to All tasks or only Completed or only Failed tasks

- 'Date Range' filter to view task as per the following:

A 'Failed' task shows additional error details about what led to the failure of that task.
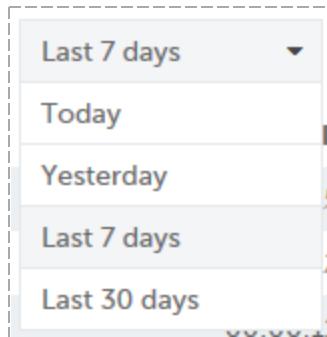
TASK NAME	DURATION	DATE	STATUS	
Extract Regular Data.atmx	00:00:03	10/10/2014 10:07:06 AM	Failed	Hide

Error: Internet Explorer Taking too long time to load the page.
An error occurred at line number 1 of the task 'Extract Regular Data.atmx'. Please open the task in the Task Editor to view the action at line number 1

Frequently Asked Questions

1. How does Anywhere Monitor work?
   Computer sends task status information to Task Status widget after every task execution.

2. Does this mean that Automation Anywhere collects information about all products on my computer?
   No.

3. Will I be contacted or receive spam based on task status information?
   No spams. However, Automation Anywhere may contact you with follow-up questions to better understand your usage of Automation Anywhere products.

4. If I choose to enable Anywhere Monitor, can I opt out later?
   Anywhere Monitor is enabled by default. If you are a Automation Standard/Premier user, you don't get any option to opt out. For Small Business/Enterprise users, this feature can be turned off by the administrator for all the clients. However, please note that after opting out of Anywhere Monitor, you will not be able to make use of the Task Status widget.

5. What information is sent to Automation Anywhere servers for Task Status widget?
   Apart from collecting the Task Status information (like task name,task duration,date run,status and error) for the widget, we also collect some secondary information like machine and location details. This secondary information is very basic and is equivalent to the information that is collected when you visit a website through a browser.

6. Can I review the information before it is sent?
   At the moment, no. This is because majority of the automation tasks are run unattended. Introducing any manual intervention for such a review would defeat the very purpose of automation and delay the task.

Live Assist

Available on purchase of Chat license only

Chat with our online support executives when you require help while using our product .

Live Assist, managed by Automation Anywhere, is one of the most advanced features that the product has to offer. If you are stuck at a point wherein you have exhausted all options of figuring out a solution to a problem and you require assistance, begin a chat with our online Support Executives, who will answer your queries, anytime.

> Note: You can avail these benefits only when you login with your Automation Anywhere Account.

Where do I locate Live Assist?

- On Login
- On the Main Client Dashboard
- While creating tasks in the Task Editor
- While configuring Commands
- While scheduling tasks in Schedule Manager
- While creating processes in Workflow Designer
- While setting triggers for tasks in the Trigger Manager
- When managing Reports and Repository

How do I connect to Live Assist?

To be connected to Live Assist you must login to your Automation Anywhere Account:

You are connecting to Automation Anywhere Live Chat!

Login to your Automation Anywhere Account

john.smith@automationanywhere.com

●●●●●   Forgot?

Login

Create an account

Benefits of Login:

By proceeding, you agree to our terms & conditions

On login, you will be attended to by one of our Chat Executives:

Have a Question?

info:

You are now chatting with 'Parthiv'

Parthiv:

Hello John.

Parthiv:

How may I assist you?

John.Smith:

Hello Parthiv.

John.Smith:

I would like some assistance in migrating to a higher version. Can you guide me through the process please?

Parthiv:

Definitely.

Type your message here

Submit

- To send your conversation(s), click Submit
- To Pin the chat window, click
- To end your chat session, click

For reference, you can opt to receive a copy of your conversation and also rate your experience:

AUTOMATION
ANYWHERE
Go be great.

Have a Question?

Thank you
for chatting with us!

Would you like us to email you a copy of this chat?

◉ Yes   ○ No

If yes, please provide your email:

| john.smith@automationanywhere.com |

How would you rate your experience with Automation Anywhere?

⊖ ★★★★★

How would you rate your chat representative today?

⊖ ★★★★★

| Your Comments |
| |

Send

Anywhere Support

 Search online support for help while using the product.

The Automation Anywhere Support site is an interactive hotspot for users with Automation Anywhere Account to seek answers to queries, participate in discussions and suggest features that they believe would enhance their automation experience.

> Note: You can avail these benefits only when you login.

You can either login through the product - Enterprise, Small Business, Premier or Standard; via search in all clients; your Automation Anywhere Account and even from the website.

You benefit from logging in as apart from searching for topics in the online knowledge base, you can:

- View demos
- Download Sample Tasks
- Refer command examples
- Post comments
- Suggest Features
- Stay connected to the Automation Anywhere community
- Participate in forum discussions
- Analyze Case Studies, White Papers and Articles
- Read customer testimonials
- Submit a request for assistance
- View your open/closed and recently solved requests

Where to Seek Help?

- On Login
- Working within the Client, Control Room or Server, as the case be.
- Creating tasks in the Task Edit
- While Scheduling tasks in Schedule Manager
- While creating processes in Workflow Designer
- While setting triggers for tasks in the Trigger Manager
- While configuring Commands

Other User Resources: User forums   Ask the Expert   Open a support case   Automation Examples

Creating an Automation Task

These topics describe how you can create automated processes or *tasks* in an efficient manner.

In this Section

- Recording an Automation Task
    - Using the Standard Recorder
    - Using the Web Recorder
    - Using the Object Recorder
- Creating a Task Using the Task Editor

Recording an Automation Task

Automation Anywhere provides several ways to create automation tasks. To ensure that you create a reliable task, you'll want to select the right method.

When planning your automation task, be sure to select the Recorder (or Task Editor) that best matches the behavior of your task.
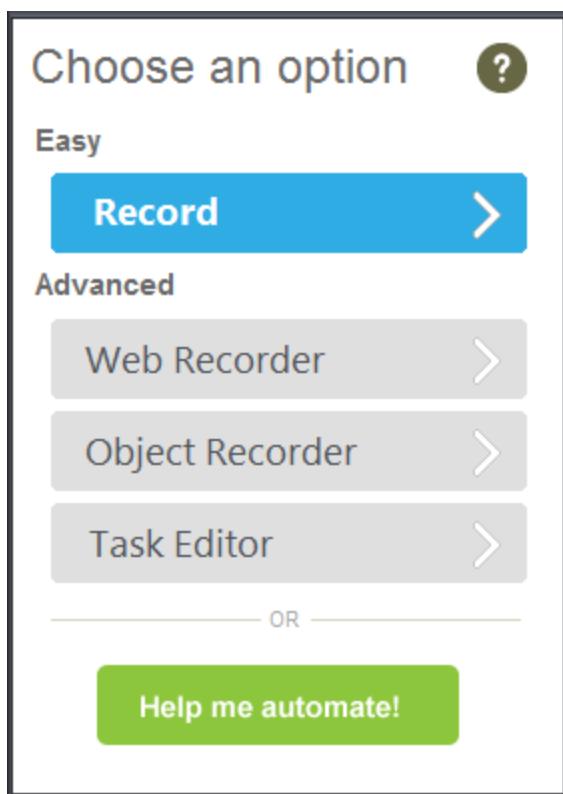
Selecting a Recording Method

In most cases, the recorder you select will be based on the type of process that you want to automate. For example, for a process that is based on web applications to extract data from websites, you would select the Web Recorder.

To record a new task, click New button at the top of the client screen.

A window is displayed that enables you to select from several methods for creating a task:

(Image as seen from version 7.5 onward)

- **Record -** It provides the easiest way to create simple automation tasks. After recording a task using the Standard Recorder, you can make changes to it by using the Task Editor.

  - In most cases, recording and editing a task is much faster than creating the task manually using the Task editor.

    > 📝 Note: You can edit the recorded tasks using the Task Editor. Refer '**Using the Task Editor**' for details.

- **Web Recorder -** It understands all web controls. Hence applicable when recording web-only tasks. If website controls on a page move location, the program can locate the correct controls and adjust the task to run without error. If the properties of the web controls change, the automation task can be updated to ensure that the task runs successfully. Refer Using the Web Recorder for details

- **Object Recorder -** Use it to capture Windows control actions such as text in drop-down menus, list boxes, and buttons; the status of radio buttons and check boxes, click actions, capturing tree views, and more. Refer Using the Object Recorder for details.

- **Task Editor -** Use it to create tasks manually. You can assign relevant actions to the task using the commands provided in the editor. Refer Using the Task Editor for details.

Also See...

- Using the Standard Recorder
- Using the Web Recorder
- Using the Object Recorder
- Using the Task Editor

Using the Web Recorder

If you want to automate processes (tasks) that involve a web application or browser, use the Web Recorder. It understands all web objects and controls, such as:

- URL command line
- Drop-down menus and command bar icons
- Web page text fields, graphics, and buttons
- Navigation and hotspot links
- Forms

> Note: If the layout of a web page changes and the web controls move to a different location on the page, Automation Anywhere is capable of locating the new position of these controls and running the task correctly.

Common Use Cases

Most people use the Web Recorder to perform the following actions:

- Repeatedly extracting data from multiple web pages
- Extracting data from tables on web pages
- Web form filling

Use the Web Recorder when you want to automate the following processes (tasks):

- Opening web pages
- Logging into a website
- Navigating through search
- Entering data into forms and submitting them
- Extracting data and updating database records
- Using a Web-based ERP (Enterprise Resource Planning) system
- Extracting web data to a local file
- Testing an online application

Also See...

- Recording Web Actions with Web Recorder
- Editing a Task with Web Recorder Command
- Extracting Data from Web Sites
- Usage Scenarios: Using Web Recorder

Recording Web Actions with Web Recorder

The web recorder is basically used to record actions for web-only applications.
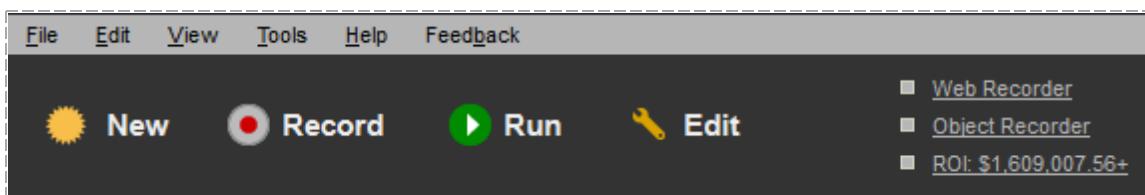
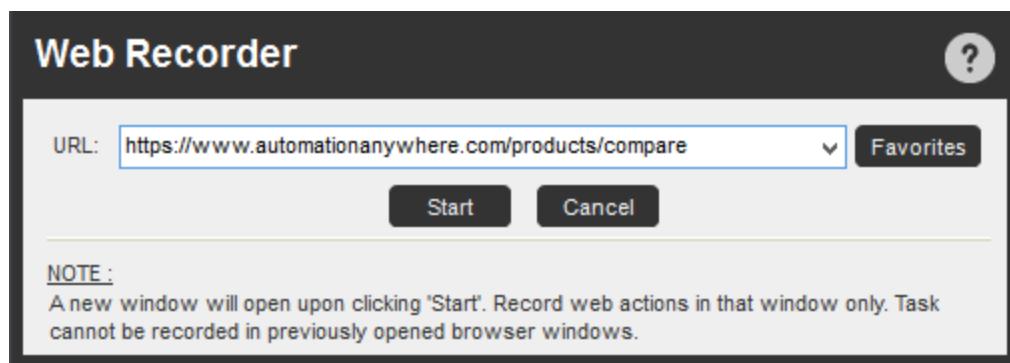To use the Web Recorder to record actions on a web site, follow these steps:
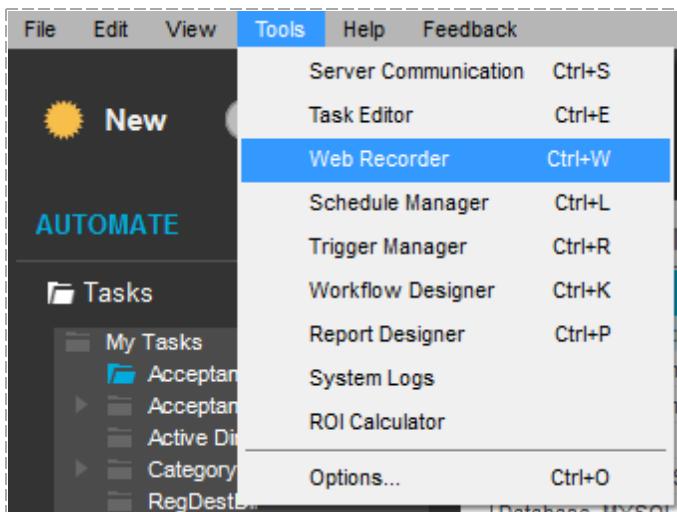
1. In the Automation Anywhere main window, launch the Web Recorder in one of the following ways:
   - Click New Task button and select Web Recorder.
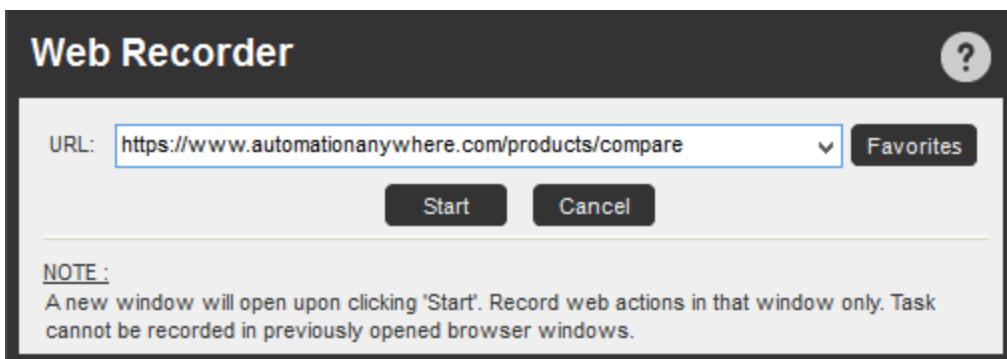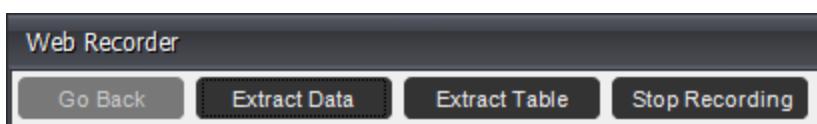
   - Or click on the Web Recorder link at the top.

   - Or click on the Tools menu and select Web Recorder.

   The initial Web Recorder window is displayed.
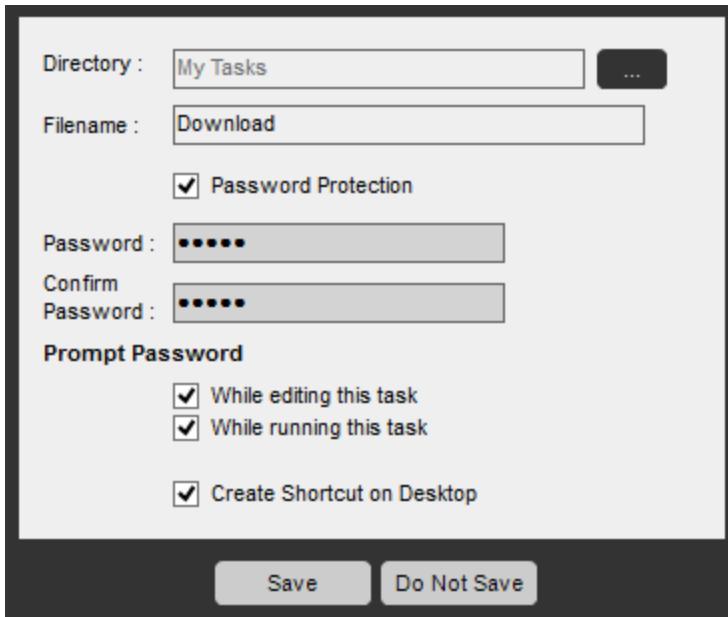
1. Enter the website address (URL) on which you want to record actions, and click Start. You can add the URL to your Favorites for easy access in the future.

The Web Recorder Toolbar is displayed, and Internet Explorer is launched in a new window with the website.

2. Perform your actions on the website, including clicking on buttons, clicking on links, and typing in text.

3. Click the Stop Recording button on the Web Recorder toolbar.

4. In the Save Task window, specify a task name, optionally select a folder in which to save the task, and click on Save. You can also optionally specify a password to prevent the task from being run by unauthorized users.

> Note: You will need to purchase the 'Password Protection' functionality for it to be available.

6. To run the task, double-click on the task, or select it and click on Run button.

7. To stop a running task, press and hold the escape (ESC) key for 2-3 seconds. You can also configure your own hotkeys to stop tasks. Refer Defining Client Hotkeys for details.

> Note: When you edit the task using the Task Editor, the program displays the Web Recorder commands and objects that were used while recording the task.

Also See...

- Extracting Regular Web Data
- Extracting Pattern-Based Data
- Extracting Table Data

Extracting Data from Websites

Extracting data from web pages can take one of four forms, depending on the type of data that requires extracting:

- **Regular data:** Unstructured data that has no repetitive patterns. An example is a paragraph of text.
- **Pattern-based data:** Structured data that repeats a pattern, such as a row in a table or a list.
- **Table data:** Data that resides in a table.

Also See...
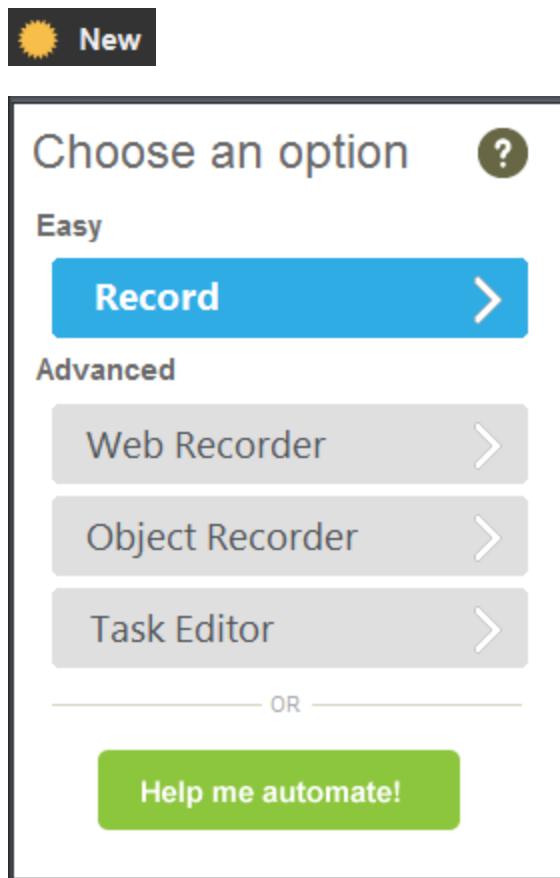
- [Extracting Regular Web Data](#)
- [Extracting Pattern-Based Data](#)
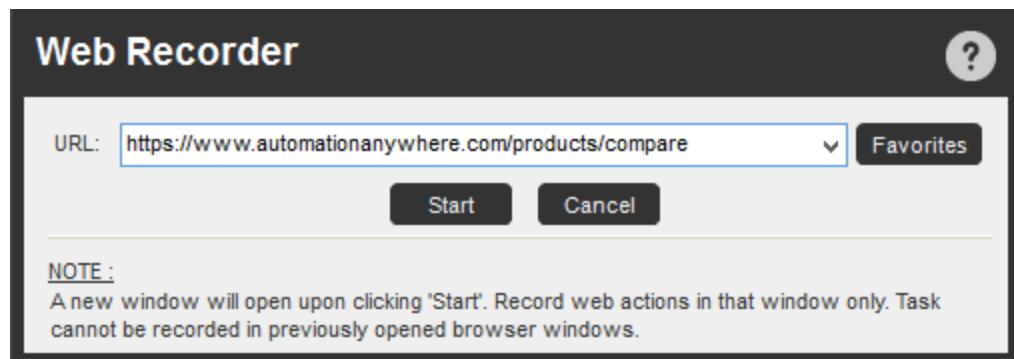- [Extracting Table Data](#)

Extracting Regular Web Data

To extract regular data from a web site, follow these steps:

1. In the Automation Anywhere main window, launch the Web Recorder in one of the following ways:

    - Click the New button and select Web Recorder.

    - Or click on the Web Recorder link at the top.
    - Or click on the Tools menu and select Web Recorder.

    The initial Web Recorder window is displayed.

2. Enter the website address (URL) on which you want to extract data, and click Start.

    The program displays the 'Web Recorder Tip' window:

While recording your task using web recorder, please wait for the web page to completely load before performing any action in it.
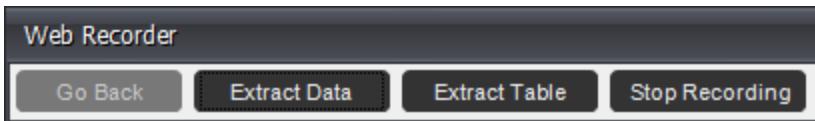
☐ Don't show this message again   OK

> **Note:** Select 'Don't show this message again ' if you do not wish the program to display the tip.

The Web Recorder Toolbar is displayed, and Internet Explorer is launched in a new window with the website.

Web Recorder

Go Back   Extract Data   Extract Table   Stop Recording

> **Note:** This feature works with Internet Explorer only.

3. You can now perform any actions you like, including mouse clicks and text entry. When you need to extract regular data from the website, click the Extract Data button on the Web Recorder toolbar.
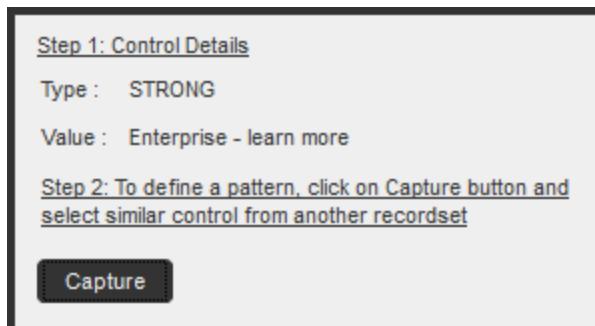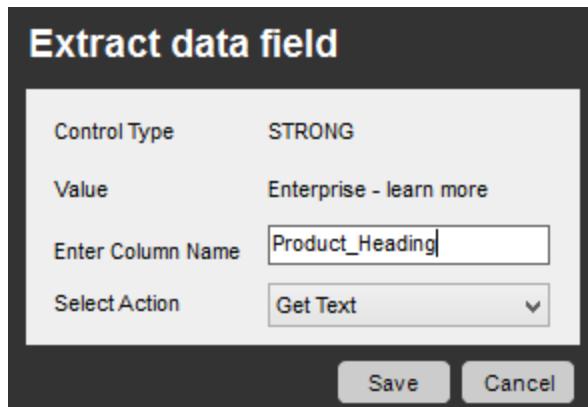
The Extract Data Option window is displayed.

Extract Data   ?

◉ Regular data
○ Pattern based data

                    Next   Close

4. Select the Regular Data radio button and click Next.

5. You can now highlight the data you want to extract.
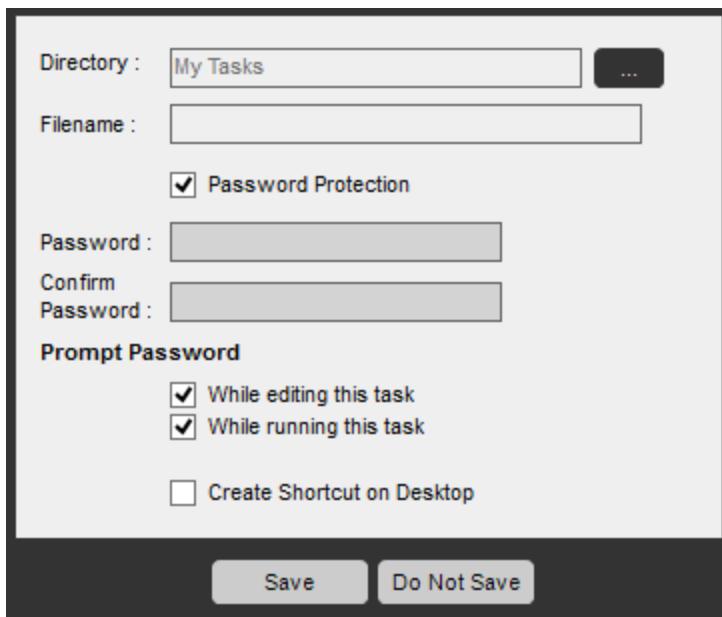
Please click on the data you want to extract. Press 'Esc' to cancel and continue recording.
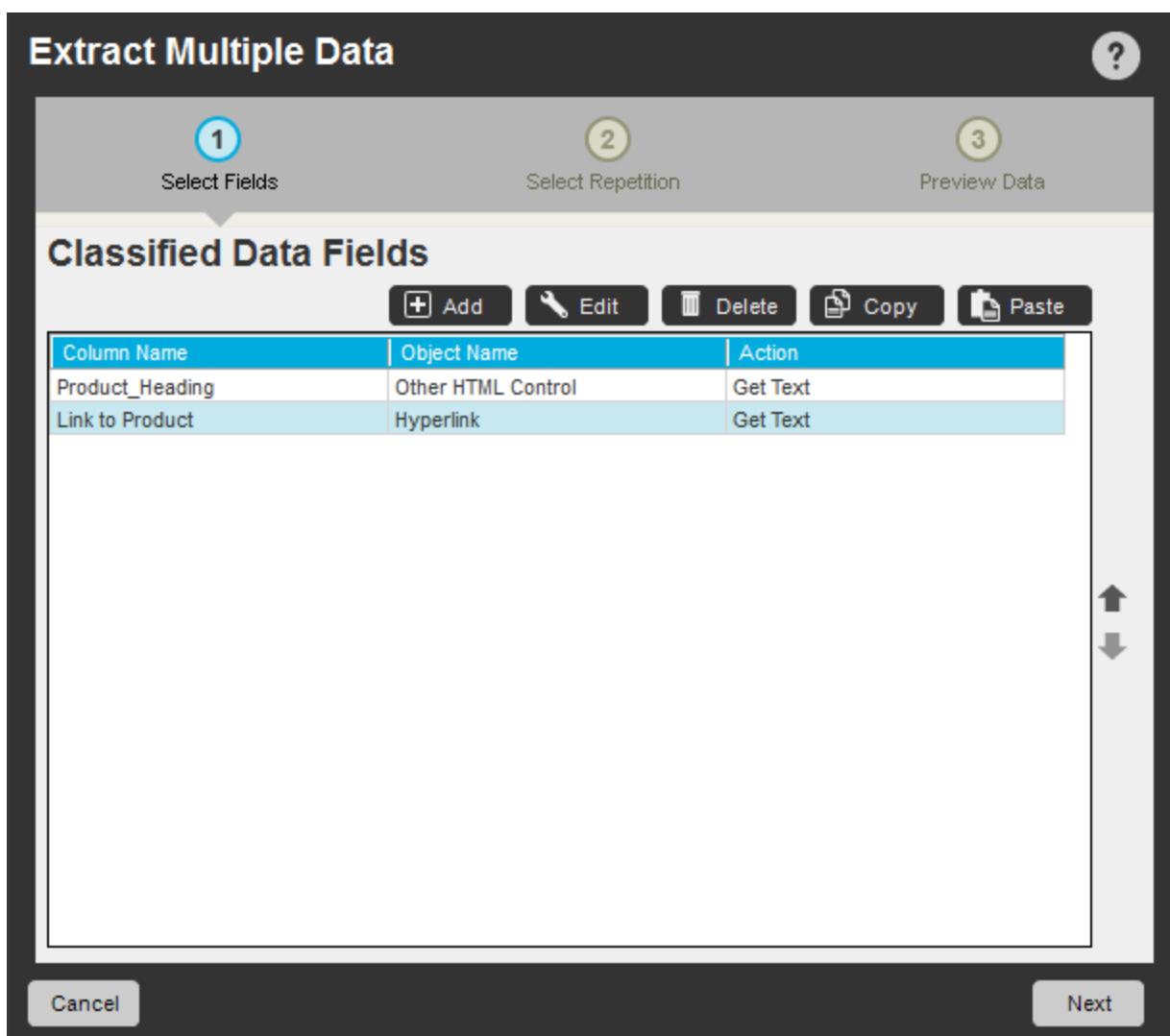
6. After highlighting the text, the Extract Data window is displayed, and contains the text you selected.

6. Save this copied text to an existing variable in your task.

7. Optionally, save the control URL either as a new variable or to an existing variable in your task.

8. You can also assign Before and After Keywords for extracting data.

9. Click Save.

10. Click the Stop Recording button on the Web Recorder toolbar.

11. The Save Task window pops-up, wherein you can specify a task name. Optionally select a folder in which to save the task and click save. You can also optionally specify a password to prevent the task from being run by unauthorized users.

12. Click Save.

💡 Tip:

> 1. To stop a running task, press and hold the escape (ESC) key for 2-3 seconds.
>
> 2. You can also configure your own hot keys to stop tasks.

Also See...
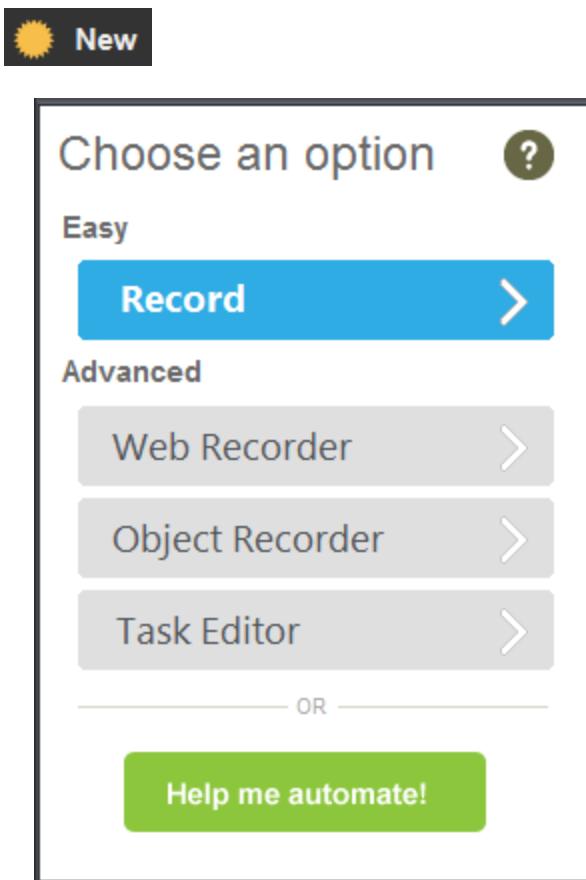
- Extracting Pattern-Based Data
- Extracting Table Data

Extracting Pattern-Based Data

When you need to extract data, which for instance, is spread over multiple pages and contains elements such as links you can use the 'Pattern Data' option in Extract Data .

To extract pattern-based data from a web site, follow these steps:

1. In the Automation Anywhere main window, launch the Web Recorder in one of the following ways:
    - Click the New button and select Web Recorder.

        New

        Choose an option ?

        Easy

        Record >

        Advanced

        Web Recorder >

        Object Recorder >

        Task Editor >

        —— OR ——

        Help me automate!

    - Click on the Web Recorder link at the top.
    - Click on the Tools menu and select Web Recorder.

        The initial Web Recorder window is displayed:

        Web Recorder ?

        URL: https://www.automationanywhere.com/products/compare ▾ Favorites

        Start   Cancel

        NOTE :
        A new window will open upon clicking 'Start'. Record web actions in that window only. Task cannot be recorded in previously opened browser windows.

2. The program displays the 'Web Recorder Tip' window:

While recording your task using web recorder, please wait for the web page to completely load before performing any action in it.

☐ Don't show this message again   OK

> Note: Select 'Don't show this message again ' if you do not wish the program to display the tip.

The Web Recorder Toolbar is displayed, and Internet Explorer is launched in a new window with the website.

Web Recorder

Go Back   Extract Data   Extract Table   Stop Recording

> Note: This feature works with Internet Explorer only.

3. You can now perform any actions you like, including mouse clicks and text entry. When you need to extract regular data from the website, click the Extract Data button on the Web Recorder toolbar.

The Extract Data Option window is displayed.

Extract Data   ?

○ Regular data
◉ Pattern based data

Next   Close

4. Select the Pattern-based Data radio button and click Next.

5. You can now highlight the data you want to extract

Please click on the data you want to extract. Press 'Esc' to cancel and continue recording.

6. Use it to set a pattern:

    a. Highlight the first line of text and click on Capture in the window that pops up:

Step 1: Control Details

Type :   STRONG

Value :   Enterprise – learn more

Step 2: To define a pattern, click on Capture button and select similar control from another recordset

Capture

b.   Highlight a second line of text. This will set the pattern. The Extract Data Field window is displayed.

Extract data field

Control Type	STRONG
Value	Enterprise – learn more
Enter Column Name	Product_Heading
Select Action	Get Text

Save   Cancel

c.   Enter a name in the "Enter Column Name" field and click Save.

- The Extract Multiple Data window is displayed, containing the column name you saved.

Extract Multiple Data   ?

① Select Fields   ② Select Repetition   ③ Preview Data

Classified Data Fields

⊞ Add   ✎ Edit   🗑 Delete   📄 Copy   📋 Paste
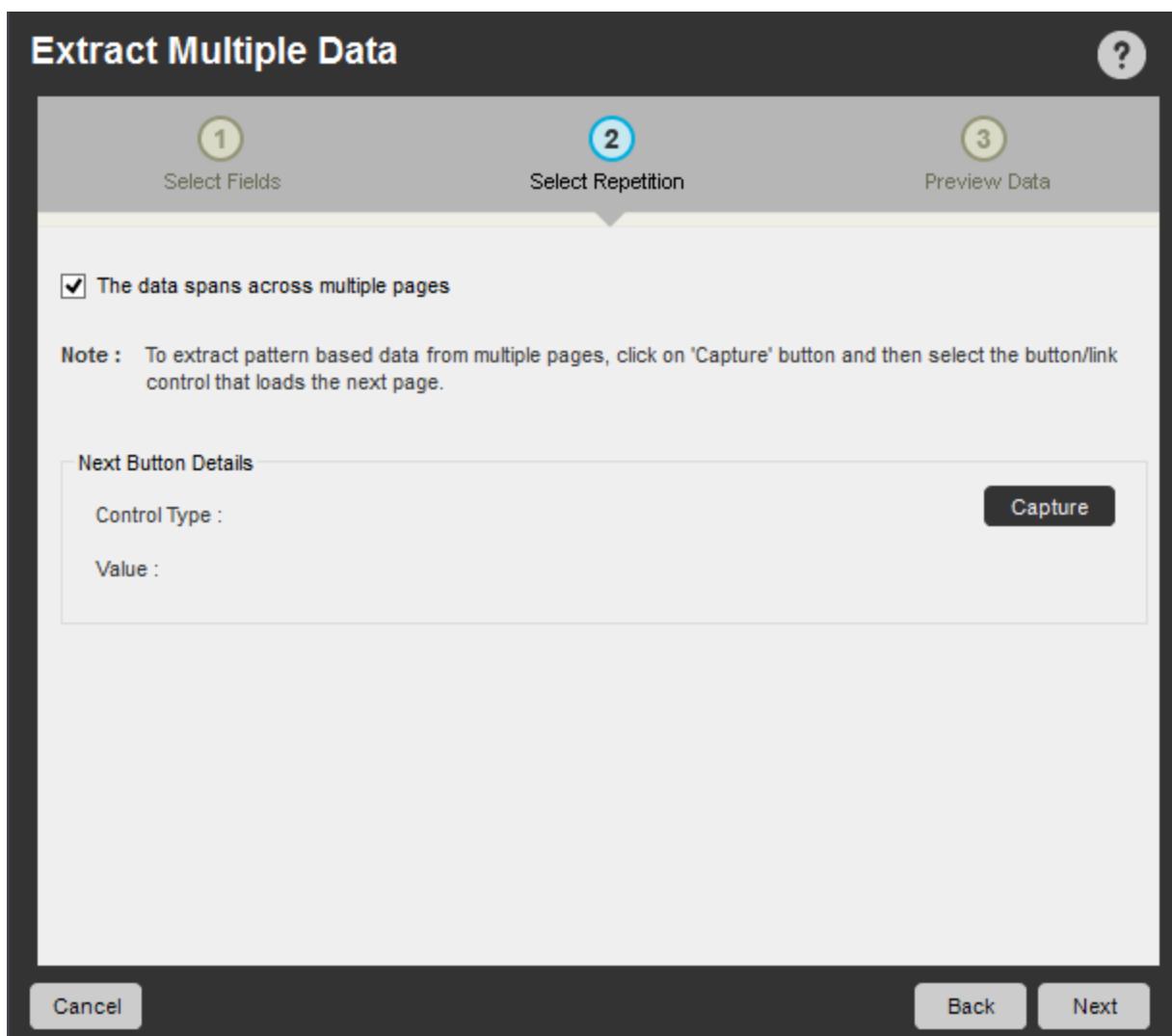
Column Name	Object Name	Action
Product_Heading	Other HTML Control	Get Text
Link to Product	Hyperlink	Get Text

Cancel   Next

In the Extract Multiple Data window, you can do several things:

- Add: Click the Add button to select additional data from the website and add another pattern that you want to capture. The data fields will be automatically viewed in the Classified Data Fields.
- Edit: Click the Edit button to change the name of the pattern.
- Delete: Click the Delete button to delete one or more patterns.

- Copy: Click the Copy button to copy one or more patterns.
- Paste: Click the Paste button to paste one or more copied patterns.
- Click Next when done.

  The second window of Extract Multiple Data is displayed:

- If the website has multiple pages from which you want to extract the patterned data, click the check box: "The data spans across multiple pages".
- Use the Capture button to capture the control name for moving to the next page. Images extracted from multiple web pages are saved in different folders.
  - Click Next when done. The Preview Data section is displayed:

- The Preview Data section displays a preview of the captured data. Save extracted images or data as CSV files in separate folders, as required.
- Click Finish.
- Click the Stop Recording button on the Web Recorder toolbar.
- The Save Task window pops-up, wherein you can specify a task name. Optionally select a folder in which to save the task and click save. You can also optionally specify a password to prevent the task from being run by unauthorized users.

Note: The trial version supports extracting 5 pages of data for viewing only. No such limitation exists in the purchased version.

Also See...
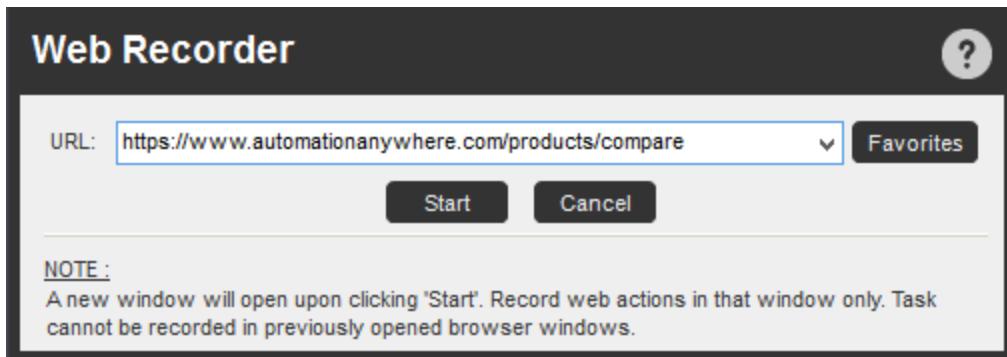
- Extracting Regular Web Data
- Extracting Table Data

Extracting Table Data

When you need to extract data from tables on web pages, the Web Recorder guides you with visual steps.

Many websites have data organized in HTML tables. Extract Table makes it easy to save tabular data in a CSV (comma-delimited) file that you can open in most spreadsheet and database applications, including Excel and Access.

1. Launch the Web Recorder:
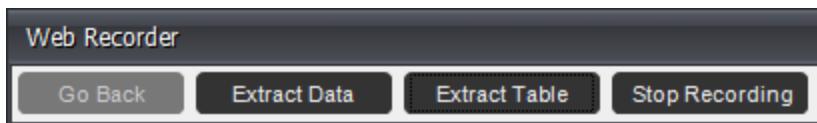
Web Recorder

URL: https://www.automationanywhere.com/products/compare   Favorites

Start   Cancel

NOTE :
A new window will open upon clicking 'Start'. Record web actions in that window only. Task cannot be recorded in previously opened browser windows.

2. Enter the website address (URL) on which you want to extract data, and click Start.

    The program displays the 'Web Recorder Tip' window:

    While recording your task using web recorder, please wait for the web page to completely load before performing any action in it.

    ☐ Don't show this message again   OK

    📝 Note: Select 'Don't show this message again ' if you do not wish the program to display the tip.

    The Web Recorder Toolbar is displayed, and Internet Explorer is launched in a new window with the website.

    Web Recorder
    Go Back   Extract Data   Extract Table   Stop Recording

3. You can now perform any actions you like, including mouse clicks and text entry. When you need to extract or copy table data from the website, click the Extract Table button on the Web Recorder toolbar.

4. Move the yellow prompt bar to the table you want to extract and click on it.

    Click on the table you want to extract from the page. Press 'Esc' to cancel and continue recording.

    📝 Note: If the content you click on is not a table, the yellow prompt bar will change to red.

    The Extracted Table window is displayed showing a preview of the data within the table.

Extracted Table

Preview (Up to 50 rows)

Sr. No.	Column1	Column2	Column3	Column4	Colu
1	Feature	Standard	Premier	Small Business	Enterp
2		$995	$2,995	$8,000	Call1-i
3		For individuals a...	For individuals lo...	For teams lookin...	For co
4	Record				
5	Task Editor				
6	Web Recorder				
7	Scheduler				
8	FTP/SFTP				
9	Multi task editing				
10	Excel Integration				
11	Web Data Extrac...	Limited			
12	Available Actions	275+	400+	530+	575+
13	Triggers				
14	Auto Login				

☐ The table spans across multiple pages

Note : To extract tabular data from multiple pages, click on 'Capture' button and then select the button/link control that loads the next page.

Next Button Details

Control Type :   Capture

Value :

Cancel   Next

5.   If the table spans multiple web pages, click the check box: "The table spans across multiple pages".

Use the Capture button to capture the control name for moving to the next page.

6. Click Next when done to preview and save the extracted table.

Extracted Table ?

Preview (Up to 50 rows)

Sr. No.	Column1	Column2	Column3	Column4	Colu
1	Feature	Standard	Premier	Small Business	Enterp
2		$995	$2,995	$8,000	Call1-8
3		For individuals a...	For individuals lo...	For teams lookin...	For co
4	Record				
5	Task Editor				
6	Web Recorder				
7	Scheduler				
8	FTP/SFTP				
9	Multi task editing				
10	Excel Integration				
11	Web Data Extrac...	Limited			
12	Available Actions	275+	400+	530+	575+
13	Triggers				
14	Auto Login				

Table No. [1]

Extract Table Data
to a csv file : [D:\My Documents\Extract Table.csv] [Browse] 💡

☑ Append to an already existing csv file

Session Name [Default] 💡   💡 **TIP :** Press F2 to insert variable

[Cancel]   [Back] [Finish]

7. Save the table as a CSV (comma-separated) file into a location of your choice. You can also use the "Append to an existing CSV file" check box to append the extracted data to an existing CSV file.

8. Click Finish.

9. Click the Stop Recording button on the Web Recorder toolbar.

10. In the Save Task window, specify a task name, optionally select a folder in which to save the task, and click on Save. You can also optionally specify a password to prevent the task from being run by unauthorized users.

Note: The trial version supports extracting 5 pages of data for viewing only. No such limitation exists in the purchased version.

Also See...

- Extracting Regular Web Data
- Extracting Pattern-Based Data

Using the Standard Recorder

You can use the Standard Recorder to record a task or a series of actions on your computer when:

- Building simple tasks for processes that involve mouse clicks and keyboard operations.
- You plan to run the automated process (the recorded task) on the same computer on which it was recorded. This ensures that the screen display coordinates are the same as they were when the task was recorded.
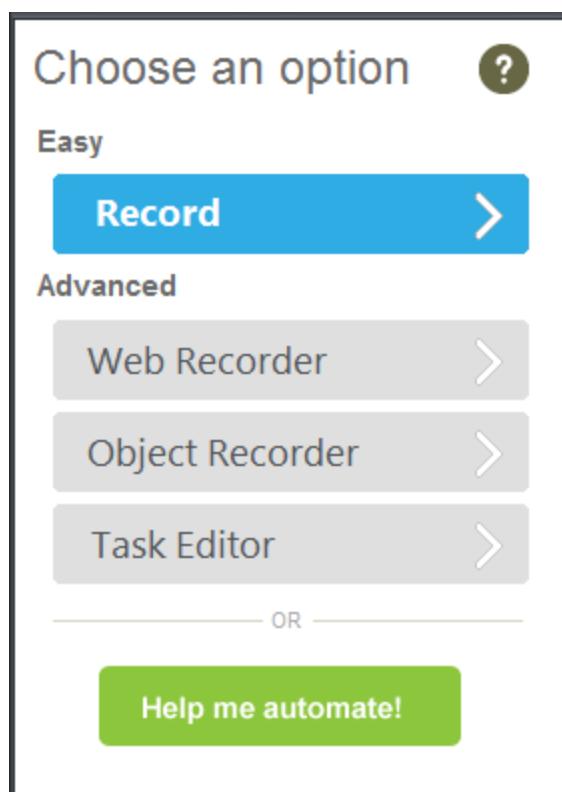
Recording a Task using the Standard Recorder

To create an automation task using the Standard Recorder, follow these steps:

1. In the main window, launch the Standard Recorder in one of three ways:

   a. Click on Record button at the top.
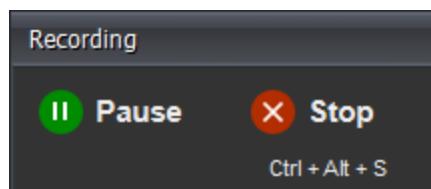
   b. Click on New Task button and select Record on the Create New Task dialog:
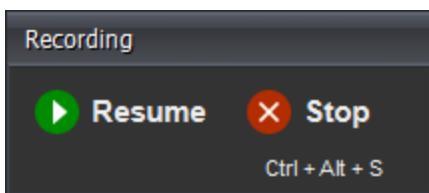
   c. Click on the File menu, and select Record.

      The Automation Anywhere window is minimized, and the Recording toolbar is displayed in the lower right corner of your computer screen.

📝 Note: If needed, you can pause the recording operation and resume it.

2. Perform all of the actions on the computer that you want to record in the task.
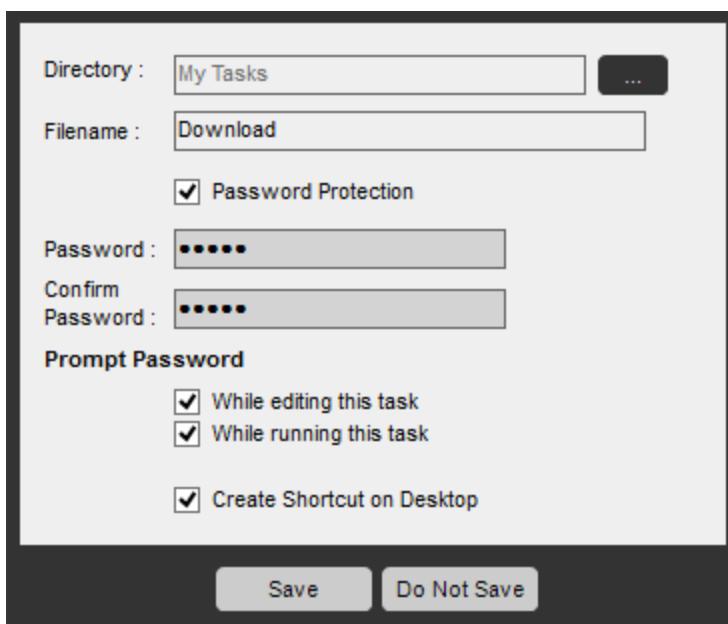
   📝 Note: While recording, the Standard Recorder recognizes the various windows that are open on your desktop.

3. When done, click on the Stop button on the Recording toolbar.

4. Save the task by specifying a name, and optionally select a folder in which to store it. When saved, the task name is displayed in the main Task List window. You can also optionally assign a password to ensure no unauthorized users run the task*.

   Password Protection functionality available with purchase version only.

   📝 Note: You can edit your recorded task using the Task Editor. Refer 'Editing a Task'.

Also See...

- Using the Task Editor
- Editing a Task
- Running a Task
- Open Program/File Command
- Manage Windows Controls Command

Using the Object Recorder
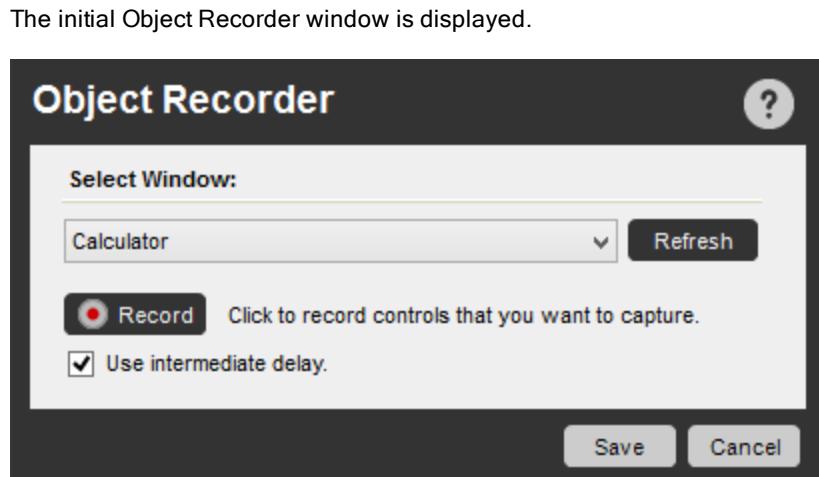
The Automation Anywhere Object Recorder is a special automation recorder that understands and captures windows objects and controls such as:

- Text from drop-down menus
- List boxes
- Buttons
- Status from radio buttons and check boxes
- Mouse click actions
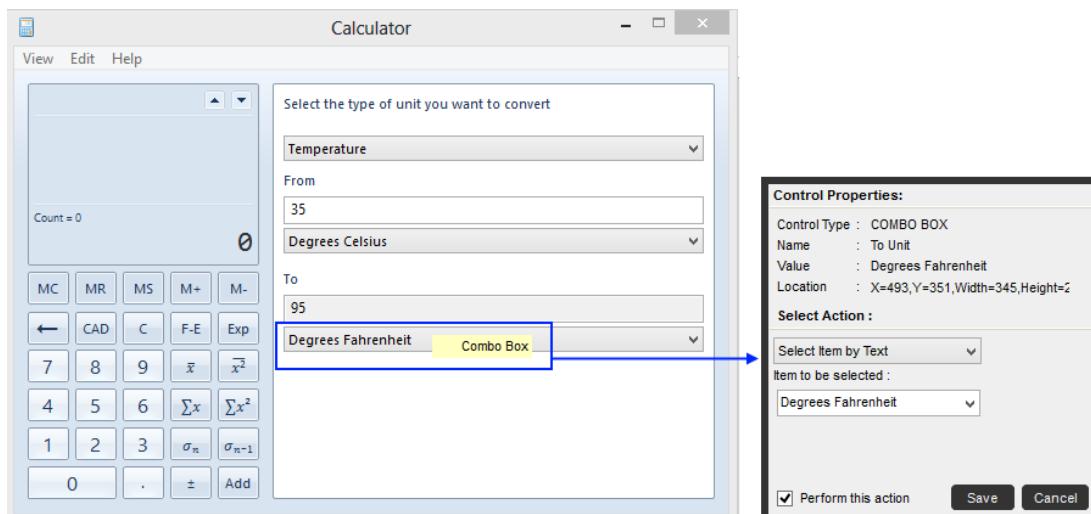
To create a new task using the Object Recorder, follow these steps:

1. In the Automation Anywhere main window, launch the Object Recorder in one of the following ways:
   - Click the New button and select Object Recorder.
   - Click on the Object Recorder link at the top.
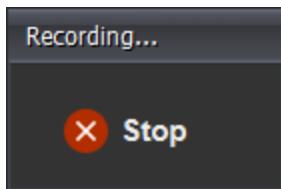
     The initial Object Recorder window is displayed.
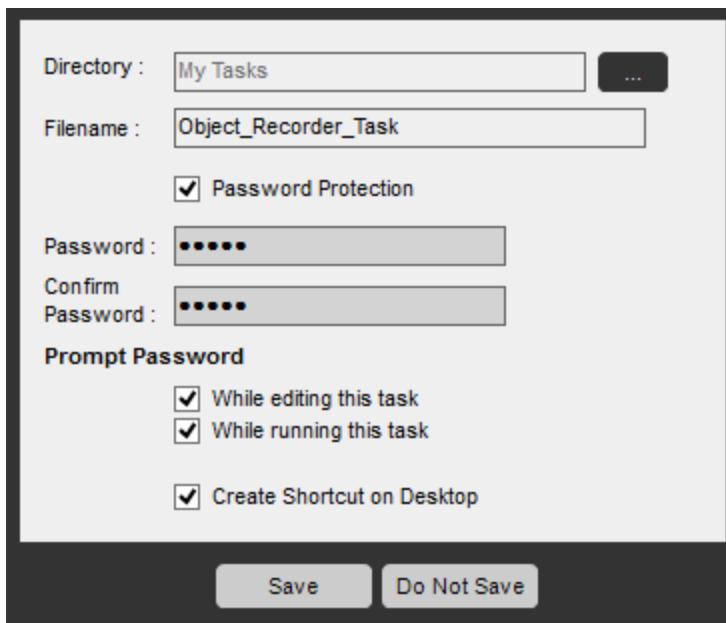
2. In the Select Window drop-down list, select the application you want to record. To perform enhanced recording, check the "Use Intermediate delay" check box.

3. Click the Record button to begin recording your actions.

4. A pop-up menu is displayed to guide you in recording. While recording, you can select a Control Action and simultaneously view the Control Properties of each action.

4. To stop recording, click the Stop Recording button.

5. In the Save Task window, specify a task name, optionally select a folder in which to save the task, and click on Save. You can also optionally specify a password to prevent the task from being run by unauthorized users.

6. To run the task, double-click on the task, or select it and click on the Run button.

You can also set the SnapPoints to view your actions, step-by-step, by using the Visualize feature. Refer Using SnapPoints for details.

Also See...

- Using the Task Editor

Creating a Task Using the Task Editor

The Automation Anywhere Task Editor is the most powerful facility in the product. Hundreds of commands and wizards are available that enable you to automate almost any process-without programming.

If you want to manually create an automated task from scratch, you can use the Task Editor.
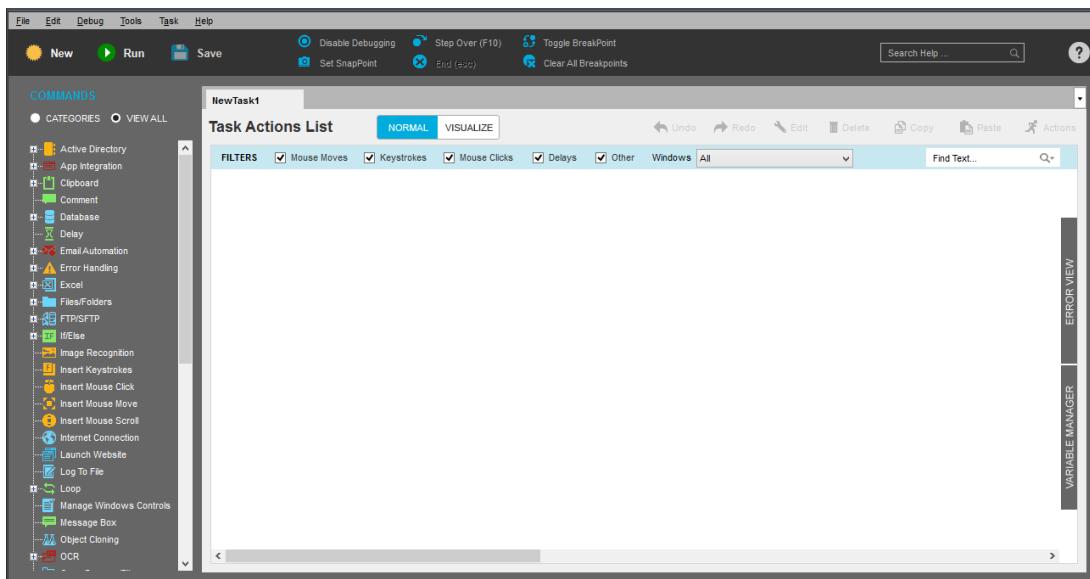
1. Launch the Task Editor by taking one of the following actions:

    - Click on the New button and select Task Editor.

    - Click on the File menu and select New, and select Task Editor.
    - Click on the Tools menu and select Task Editor.

2. Add actions to the new task by dragging commands from the left pane of the editor to the Task Actions List pane. After inserting most commands, you are prompted to specify the properties that apply to those commands.

(Image as seen in Enterprise edition version 8.2.1)

3. Save the task.

Also See...

- [Creating an Automation Task](#)
- [Using the Task Editor](#)

Other User Resources: User forums | Ask the Expert | Open a support case | Automation Examples

Using the Task List

Automation Anywhere provides a simple, visual view of your automation tasks in the Task List section. Here, you can easily navigate task folders and work with your tasks.

File Name	Created At	Hotkey	Description	Schedule	Type	Repeat	Status	Last Run Time
Database Update.wkfx	02/11/2014 15:57:04				Workflow File	Do not Repeat	Complete	02/11/2014 16:08:23
RegEx.wkfx	01/31/2014 13:10:20				Workflow File	Do not Repeat	Complete	01/31/2014 13:15:46
Run Report.wkfx	12/19/2013 17:51:57				Workflow File	Do not Repeat	Complete	12/19/2013 17:52:37
run task.wkfx	03/14/2014 12:35:44				Workflow File	Do not Repeat	Complete	03/14/2014 12:36:37
test1.wkfx	03/27/2014 18:22:01				Workflow File	Do not Repeat		
testworkflow.wkfx	12/17/2013 15:23:35				Workflow File	Do not Repeat	Complete	12/17/2013 15:27:36

My Workflow — New Workflow · Delete · Upload · Actions
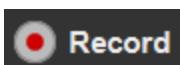
Using the buttons above the Task List in the main Automation Anywhere window, you can perform the following actions:

1. Create a new task - Record tasks with different recording methods available on the 'Create New Task' window.

   New

   Refer Selecting Recording Methods for details.

2. Record a new task - Record keystrokes as well as mouse clicks and movements.

   Record

   Refer Using the Standard Recorder for details.

3. Run the selected/existing task.

   Run

   Refer Running a Task for details.

4. Edit/modify the selected/existing task using the Task Editor.

   Edit

   Refer Using the Task Editor for details.

Using the buttons in the top-right side of the Task List, you can perform additional actions:
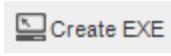
1. Delete a task.

   Delete

2. Upload a task to the Server.

   Upload

3. Convert a task to an executable (.EXE) file.

   Create EXE

4. Select an action:

- **Run:** Run the selected task (the automated process).
- **Edit:** Edit the selected task using the Task Editor.
- **Upload Task:** Upload the selected task to the Server Task Repository.
- **Copy Task:** Create a copy of the selected task.
- **Rename:** Rename the selected task.
- **Locate on Disk:** Create a copy of a task as an .atmx file in the location: My Documents\Automation Anywhere\Automation Any-where\My Tasks
- **Send To:** Send a copy of a task to: Desktop(Create Shortcut), Mail Recipient, Startup Folder or My Documents.
- **Create EXE:** Convert a task to an executable (.EXE) file.
- **Delete:** Delete a task.

Also See...

- Scheduling a Task
- Editing a Task
- Copying a Task
- Deleting a Task
- Running a Task

Editing a Task

After initially creating an automation task, you can edit the task using the Task Editor to add, modify, or delete actions and steps to the process.
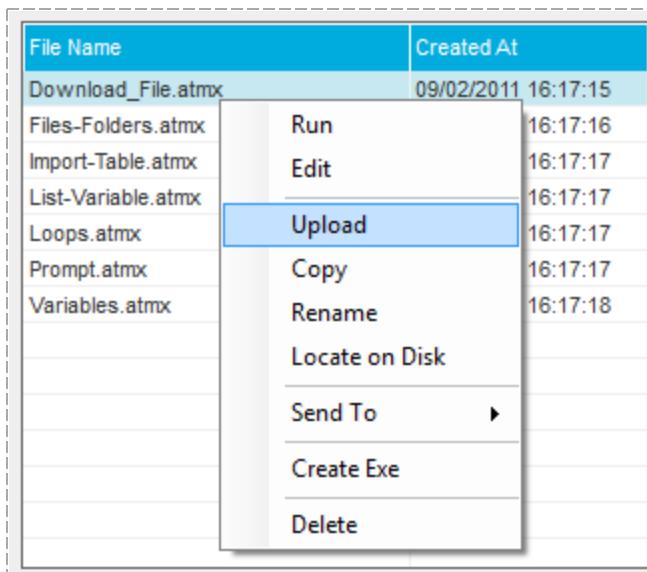
For example, you might want to remove all built-in delays to increase the speed of the task when it is run.

You can open a task in the Task Editor in the following ways:
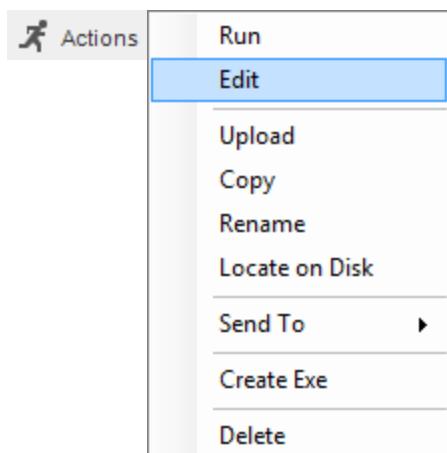
1. Select (highlight) the task and click the Edit button.

2. Right-click on the task, and select Edit from the pop-up menu.

File Name	Created At	
Download_File.atmx	09/02/2011 16:17:15	
Files-Folders.atmx		16:17:16
Import-Table.atmx		16:17:17
List-Variable.atmx		16:17:17
Loops.atmx		16:17:17
Prompt.atmx		16:17:17
Variables.atmx		16:17:18

    Run
    Edit
    Upload
    Copy
    Rename
    Locate on Disk
    Send To ▶
    Create Exe
    Delete

3. Click on the Actions button, and select Edit.

    Actions
    Run
    Edit
    Upload
    Copy
    Rename
    Locate on Disk
    Send To ▶
    Create Exe
    Delete
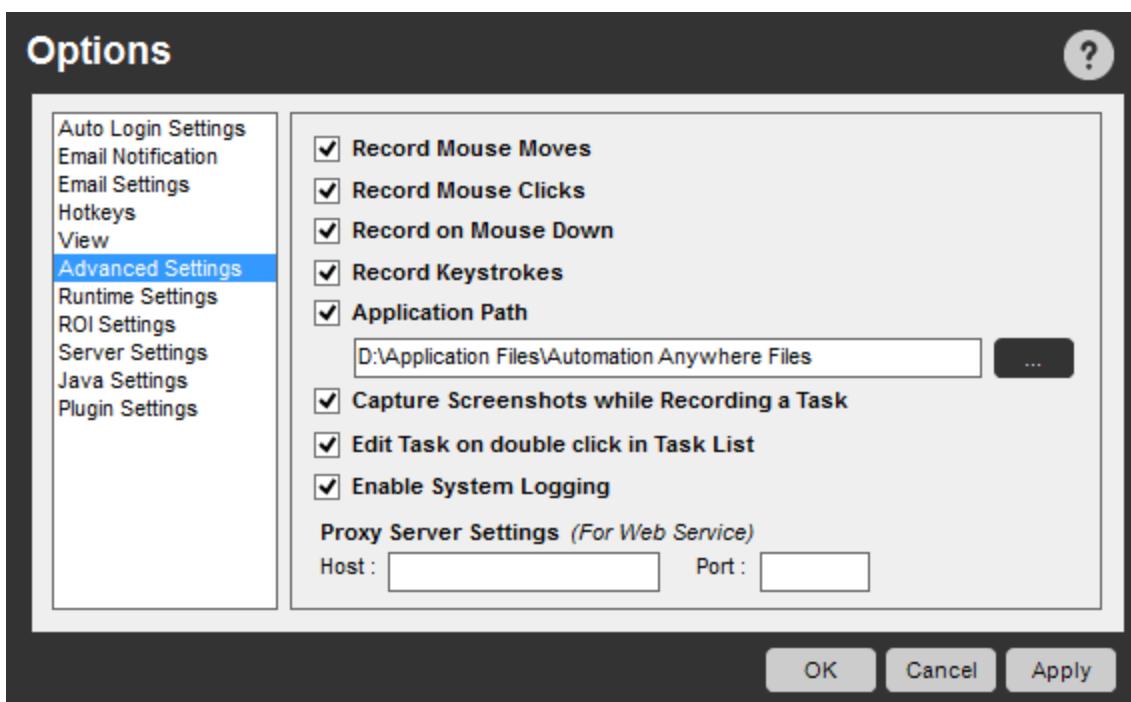
If you want to edit a task when double-clicking on it, change the setting by following these steps:

1. In the main Automation Anywhere window, click on Tools -> Options.
2. Click on Advanced Settings.

3. Check the "Edit Task on double click in Task List" check box.

Also See...

- Using the Task List
- Copying or Renaming a Task
- Deleting a Task
- Repeating a Task
- Adjusting the Speed of a Task
- Stopping a Task
- Sending Email Notifications for a Task
- Locating a Task
- Sending a Task
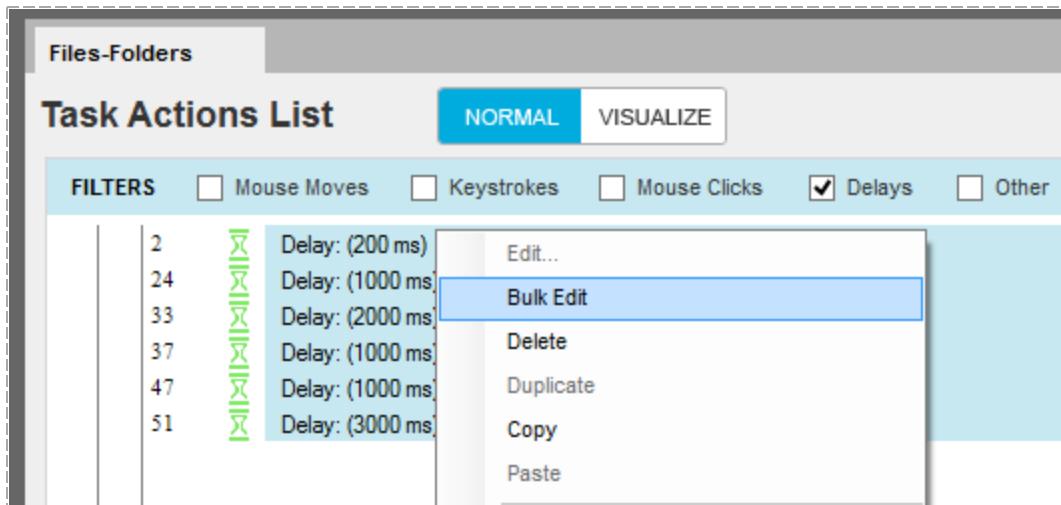
Editing Multiple Commands (Bulk Edit)

To make editing automation tasks easier, Automation Anywhere provides a Bulk Edit feature, which enables you to edit multiple commands at one time.
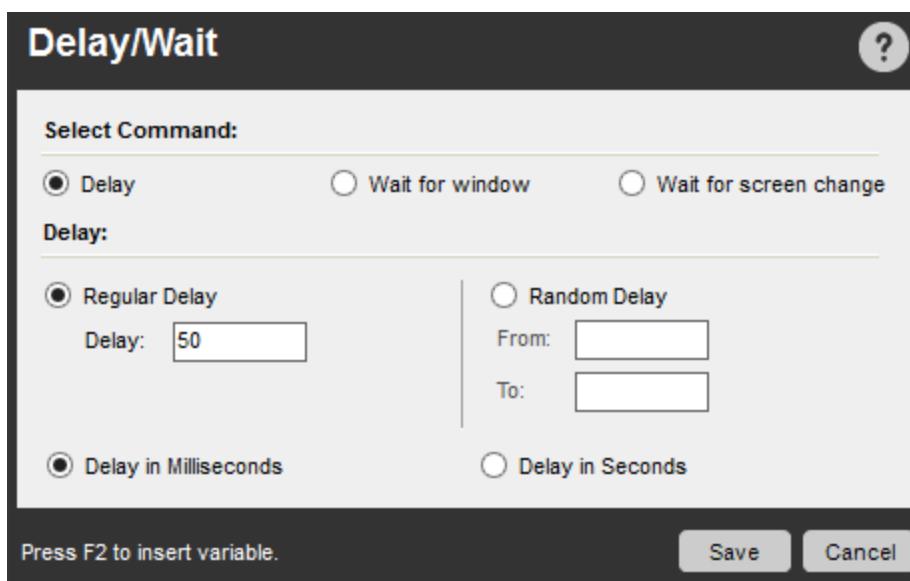
Editing Multiple Delay Commands

Using Bulk Edit, you can apply the settings for a specific delay command to multiple delay commands in your task. Follow these steps:
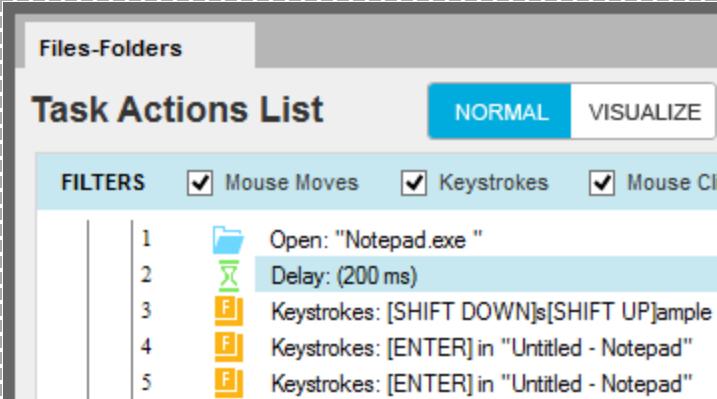
1. In the Task Editor, select multiple Delay commands in the task.
2. Right-click on one of the selected commands and select Bulk Edit from the pop-up menu.

3. In the Delay/Wait window, set values for the delay settings and click Save.

4. In the Task Actions List view of the task, you'll see that all of the Delay commands that were included in the Bulk Edit operation are now changed to the new settings.

> 📝 Note: Bulk Edit works with two command types: 'Delays' and 'Keystrokes'. You can however edit either keystrokes or delays, individually.

Use Case Scenarios

1. Set a common delay of 5 seconds to multiple Delay commands

1. Open the task in the Task Editor.
2. In the Task Actions List, select (highlight) multiple Delay commands using the Control key, or use the Filter to view all of the Delay commands in the task.
3. Right-click on one of the selected commands and choose Bulk Edit.
4. In the Delay/Wait window, enter "2000" in the Delay field.
5. Click Save.

2. Set a common random delay for multiple Delay commands

1. Open the task in the Task Editor.
2. In the Task Actions List, select (highlight) multiple Delay commands using the Control key, or use the Filter to view all of the Delay commands in the task.
3. Right-click on one of the selected commands and choose Bulk Edit.
4. In the Delay/Wait window, Select Random Delay.
5. Enter "5" in the 'From' field.
6. Enter "500" in the 'To' field.
7. Select the Delay in Seconds radio button.
8. Click Save.

Editing Multiple Keystroke Commands

Using Bulk Edit, you can apply the settings for a specific Keystrokes command to multiple Keystrokes commands in your task. Follow these steps:

1. In the Task Editor, select multiple Keystrokes commands in the task.
2. Right-click on one of the selected commands and select Bulk Edit from the pop-up menu.
3. In the Insert Keystrokes window, set values in the Keystrokes settings and click Save.

- In the Task Actions List view of the task, you'll see that all of the Keystrokes commands that were included in the Bulk Edit operation are now changed to the new settings.

Also See...

- Editing a Task
- Copying or Renaming a Task

- Deleting a Task
- Changing Window Titles

Changing Window Titles

If you record a task with an application for which the window title changes frequently, use the Task Editor commands for updating the task to refer to the proper title.

For instance, several scenarios can exist:

1. The window title of a web page or a Windows application changes continuously due to dynamic content.
   - The title in Microsoft Outlook changes frequently as the user selects new folders.
   - A web page title contains a session ID for the website.
2. Copied tasks - If you copy a recorded task to create a new task, the application titles will need to be changed.
3. When you use a wild card, the 'Current Active Window' command, or variables to use for setting window titles.

Tip: You can use the Windows filter to select a particular Window name, enabling you to view task actions that are sorted by application.This filter provides a convenient view of any or all applications that are used in your task.

How to use the Change Window Title action
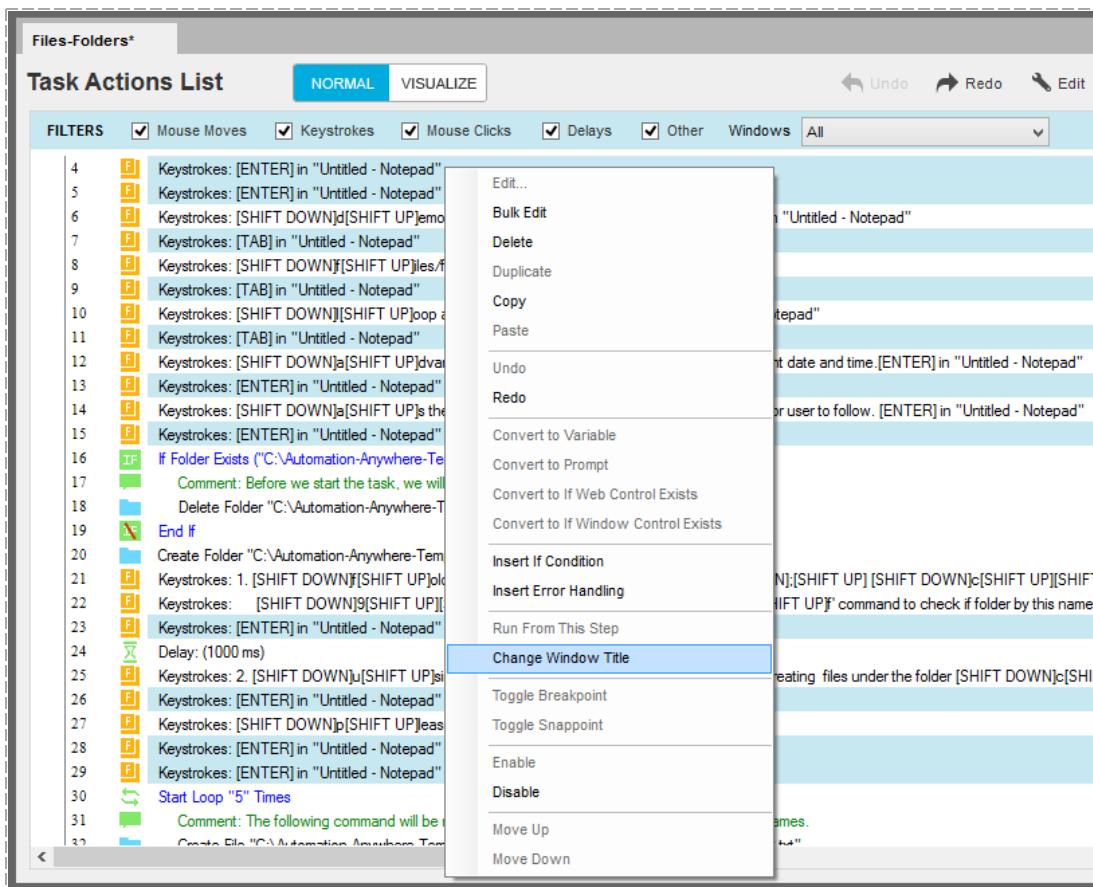
To use the Change Window Title action to apply a common window title to multiple application windows, follow these steps:

1. In the Task Editor, use the Windows filter to select a particular Window name.
2. Select one or more commands with the window title for which you want to change.
3. Right-click on one of the selected windows commands and choose the Change Window Title action. You can also use the Actions button to select the Change Window Title option.
4. Set a common Window title for the selected windows that your want to change and click OK.
   - In the Task Actions List view, you'll see that the windows titles are changed to the new title that you've assigned. Refer the example given here:
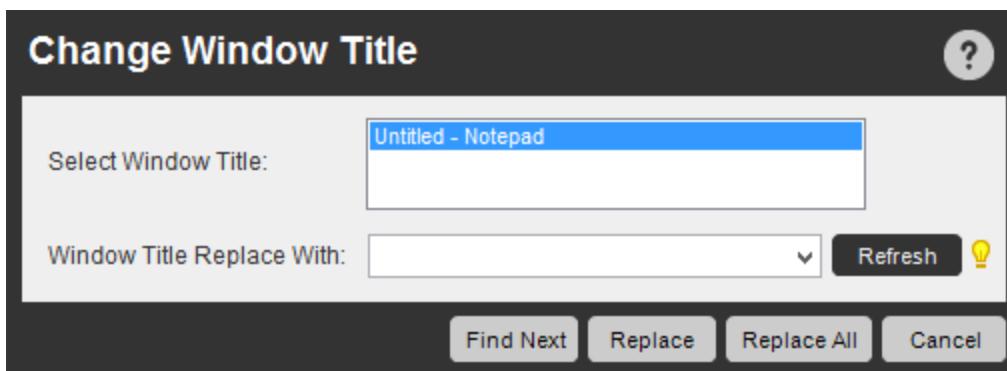
Assign a common window title to multiple window actions

1. Open the task in Task Editor.
2. In the drop-down list of the Windows filter, select one particular application window.

3. Using the Control key, select (highlight) multiple windows commands to edit their window titles.

4. Right-click on one of the selected commands, and choose Change Window Title from the pop-up menu of actions.

5. In the Change Window Title dialog, specify a window title to replace the current selected window titles.

- Use the Find and Replace commands to browse and replace the title, as appropriate.

6. Click Save.

Also See...

- Editing a Task
- Copying or Renaming a Task
- Deleting a Task
- Changing Window Titles
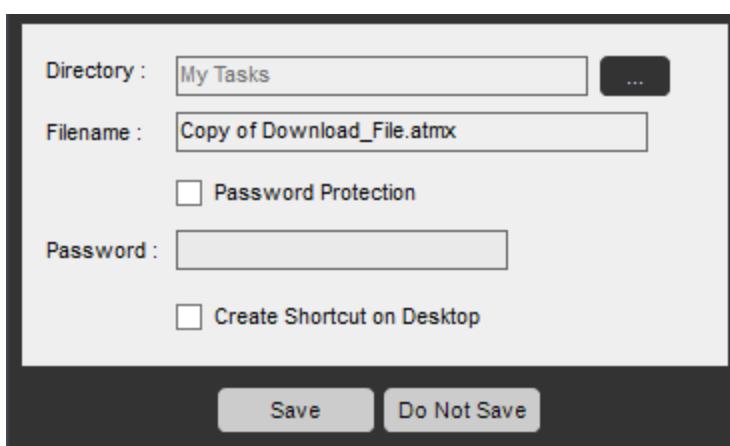
Copying or Renaming a Task

One of the enormous benefits of using Automation Anywhere is the ability to reuse processes that you automate. Copying and renaming automation tasks allows you to reuse tasks that have been previously built.

Copying a Task

To copy a task, follow these steps:

1. In the Task List, select (highlight) the task you want to copy.
2. Copy the task using one of the following methods:
- In the File menu, select Copy.
- In the Actions button, select Copy from the drop-down list.
- Right-click on the task and select Copy.
3. In the Copy Task window, select a directory location and specify a file name for the new copy of the task.
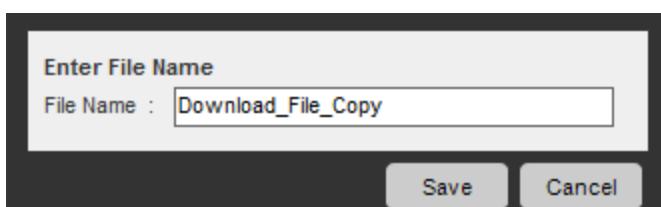
4. Optionally enable password protection for the task, and create a desktop shortcut.
5. Click Save.
- The new task is added to the Task List view.
- All commands in the original task are available in the new copy of the task.

Renaming a Task

To rename a task, follow these steps:

1. In the Task List, select (highlight) the task you want to rename.
2. Rename the task using one of the following methods:
- In the File menu, select Rename.
- In the Actions button, select Rename from the drop-down list.
- Right-click on the task and select Rename.
3. In the Rename File window, specify a new file name for the task.

4. Click Save.
- The renamed task is displayed in the Task List view.

Also See...

- Using the Task List
- Editing a Task
- Copying or Renaming a Task
- Deleting a Task
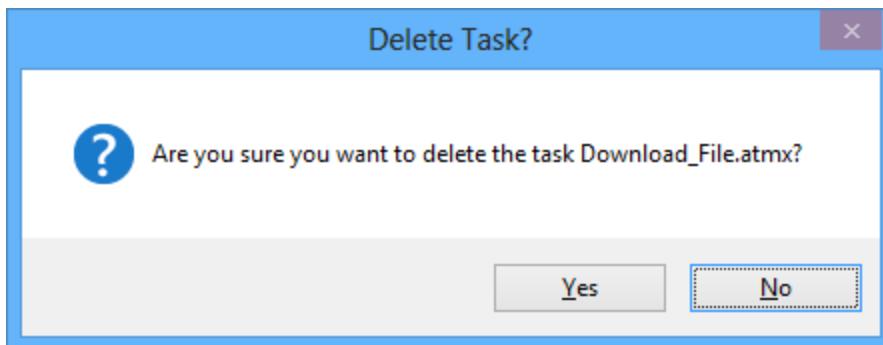
Deleting a Task

In some cases, you might want to delete an existing automation task.

To delete a task, follow these steps:

1. In the Task List, select (highlight) the task you want to delete.
2. Delete the task using one of the following methods:
   - In the Edit menu, select Delete.
   - In the Actions button, select Delete from the drop-down list.
   - Right-click on the task and select Delete.
3. In the Delete Task? window, click on Yes.

   - The task is removed from the Task List view.

> Note: After deleting a task, you cannot restore the task on the client. Before deleting a task, be sure you no longer have use for that task.

If the task was copied to the Automation Anywhere Small Business Server, you can restore it by copying the Server version to the client.

> Tip: As a best practice, be sure that all tasks are copied to the Server as a way of backing up your automation projects.

Also See...

- Using the Task List
- Editing a Task
- Copying or Renaming a Task
- Deleting a Task

Running a Task

In most cases, as you create automated tasks, you'll want to test the process to ensure that the results match your expectations. You can test an automated process by running the task.

When running tasks, Automation Anywhere provides advanced task-queuing technology with run-time settings that you can tune.

You can run a task manually or you can schedule a task to be run at a later time.

To manually run a task, follow these steps:

1. In the main Automation Anywhere window, select (highlight) the task you want to run.
2. Run the task by using one of these methods:
a. Click the Run button.
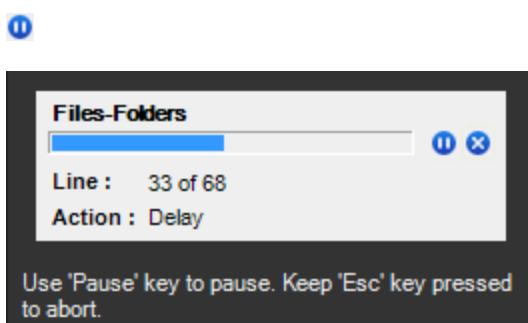
b. Click on the File menu and select Run.
c. Click on the Actions button and select Run from the drop-down list.
d. Right-click on the task and select Run.
  • When a task begins to run, a Run-Time window is displayed.
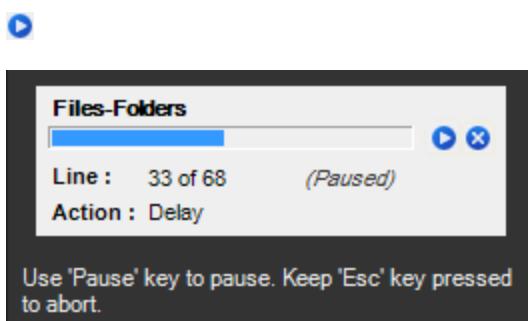
You can pause or stop a task from running at any time.

1. To pause the task, press the Pause button on the 'Run Time Window'

2. You can resume running the task by using the Play button on the 'Run Time Window'.

3. To stop a task from running, use one of the following methods:
a. Use the Stop button.
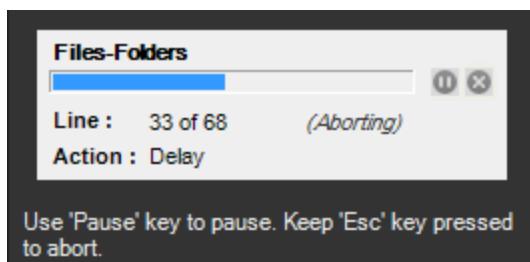
b. Press the Escape (ESC) key on the keyboard.

💡 Tip: You can also modify your hotkeys to use a different key to stop tasks that are running.

When stopping a task, the process does not end abruptly. The Run-Time window displays the message "Aborting" and completes any step that is in progress.
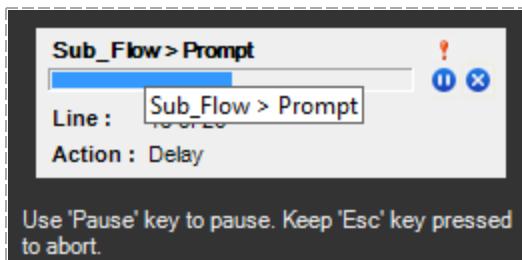
> Note: When using the 'Run Task' command, if you have a sub-task running within the main task, the window will display the task hierarchy.

The run-time window will display:

- The selected and current task being executed.
- The task's action being performed.
- The task's line number.
- Tool tip - if the task flow is too long to fit within the window.

   > Tip: Hover the cursor over the task name and the task flow is displayed as a tool tip.

Also See...

- Scheduling Tasks to Run
- Setting Up Hotkeys for a Task
- Setting Security Features for a Task

Stopping a Task

While running an automated task, you can pause or stop the process, as necessary. You can stop a task from running either manually or by inserting a conditional command within the task.

Stopping a Task Manually

To stop a task manually, simply click the Pause ⏸ or Stop ⊗ button on the progress window, or press the Escape (ESC) key on the keyboard for 2-3 seconds.

Stopping a Task Using a Condition

You can stop a task by using specific conditions by inserting a 'Stop Task' command.

A Stop Task instruction looks like this in the Task Editor:

⬛ Stop The Current Task

💡Tip: Stop the task only when encountering a file that is larger than a particular number of megabytes (MBs).

To know more about using this command refer Stop Task Command.

Also See...

- Running a Task
- Editing a Task
- Deleting a Task
- Copying a Task

Locating a Task on Your Computer

In some cases, you might want to locate the task file that is associated with an automated task you've created.

Automation Anywhere files have the file extension of: '.atmx'

To locate an Automation Anywhere task file on your computer, follow these steps:

1. In the main Automation Anywhere window, select the task in the Task List for which you want to locate the .atmx file.
2. Either click on the Edit menu or on the Actions button, and select 'Locate on Disk'. A Windows Explorer window is displayed showing the folder containing the task files.
3. Locate the automation task file. The task name matches the name you've assigned to the task.

Also See...

- Sending a Task
- Setting Task Properties
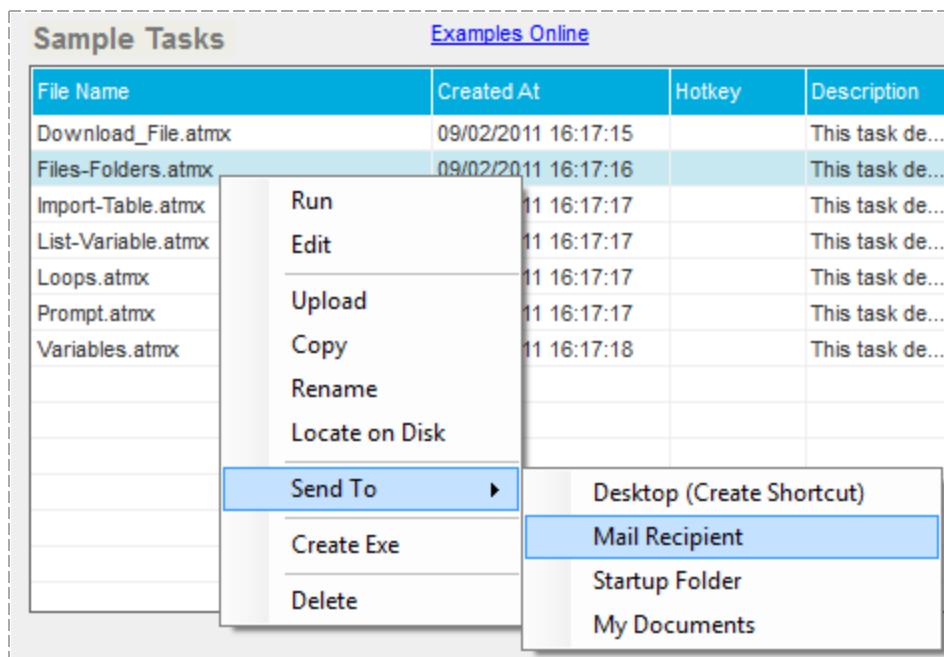
Sending a Task

Occasionally, you might want to distribute your automation tasks to other people and locations. Using the 'Send To' option, you can send a task to the following locations:

1. **The Desktop** - you can create a desktop shortcut to run the task with a click of a button.
2. **An email recipient** - Send the task to another person using an email address.
3. **The Start-up folder** - Enable the process to be run each time the computer starts by using the Windows Start-up folder.
4. **The My Documents folder** - Organize and run tasks from the My Documents folder.

To send a task to any of these locations, follow these steps:

1. Either click on the Edit menu or on the Actions button, and select 'Send To'.

2. Select one of the Send To options and complete the information, if required.
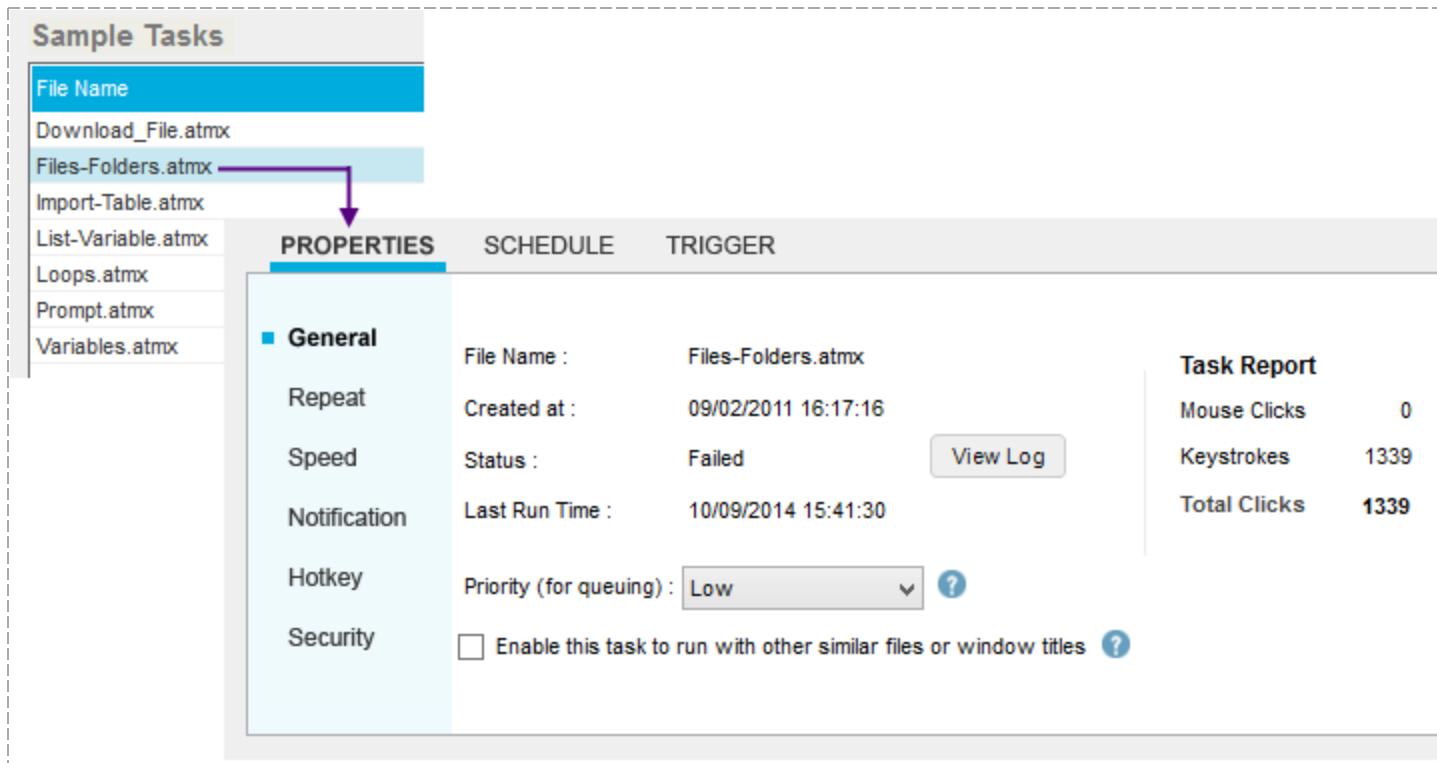
Note: In order to run a task, a resource file (.atmr) is required which is appended when you send the task.

Also See...

- Setting Task Properties
- Sending Email Notifications for a Task
- Locating a Task on Your Computer
- Scheduling Tasks to Run

ANTLETER

Setting Task Properties

After you create an automation task, you can view and change its properties.

Use the Properties tab to set and adjust the settings for your task. On this tab, you can do the following:

- View and set general properties, such as setting the priority status and viewing the date it was created.
- Specifying the frequency or conditions to repeat the task.
- Adjusting the speed for which a task runs.
- Setting up email notifications for a task.
- Specifying hot keys to use with you task.
- Setting security parameters to ensure that only authorized users run the task.
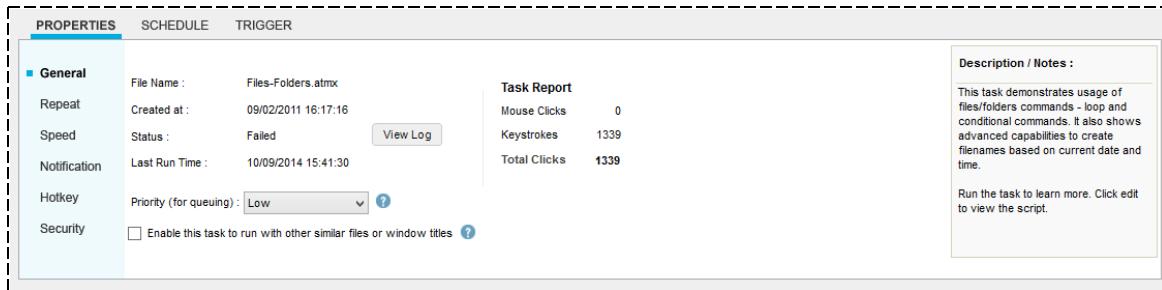
Also See...

- Viewing and Setting General Properties
- Repeating a Task
- Adjusting the Speed of a Task
- Sending Email Notifications for a Task
- Setting up Hot Keys for a Task
- Setting Security Features for a Task

Other User Resources: User forums   Ask the Expert   Open a support case   Automation Examples

Viewing and Setting General Properties

After you create a task, you can view and edit the task's general properties by using the General tab.
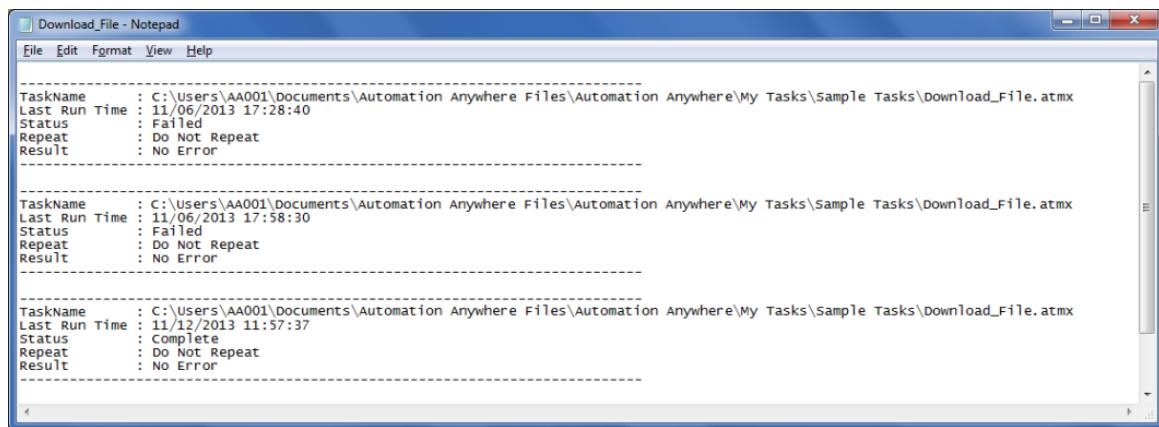
The general properties include task name, status, last run time, and other statistics. You can also use the Description/Notes field to enter useful comments about the task.

General Properties - An Overview

- **File Name** - Displays name of the task
- **Created at** - Displays the date and time of task creation
- **Status** - Displays the last time the task was run.
- **View Log** - Click View Log to see historic and detailed status of the task. If the task has failed, View Log shows the line number where the task failed.

Refer the sample of the log as displayed in notepad given below:

- **Last Run Time** - Displays the last time the task was run.
- **Mouse Clicks** - Lists the total mouse clicks recorded in a task, automatically calculated based on the task.
- **Keystrokes** - Lists the total keystrokes recorded in the task, automatically calculated based on the task.
- **Total Clicks** - Lists the total clicks (Mouse Clicks + Keystrokes) recorded in the task. (Every time you run a task, you save this many clicks and keystrokes.) Automatically calculated based on the task.
- **Priority for queuing*** - You can specify a priority for the task, which will decide the precedence of the task when it is in a queue. Also, if the priority is same then rules apply for precedence of jobs.

    * Feature also available in Standard edition version 8.1.2 or higher.

- **Enable this task to run with other similar files or window titles** - By default, the task recorded will only run on the specific file(s) it was recorded on. However, SMART Automation Technology includes an option for running the recorded task on any file within the application it was recorded in. For example, you record a task to run on the file abc.xls. By enabling this option, you can run the task on def.xls, xyz.xls, or any other *.xls file, without editing the task.
- **Description/Notes** - Add descriptions and notes for a task in the Description/Notes window.

Also See...
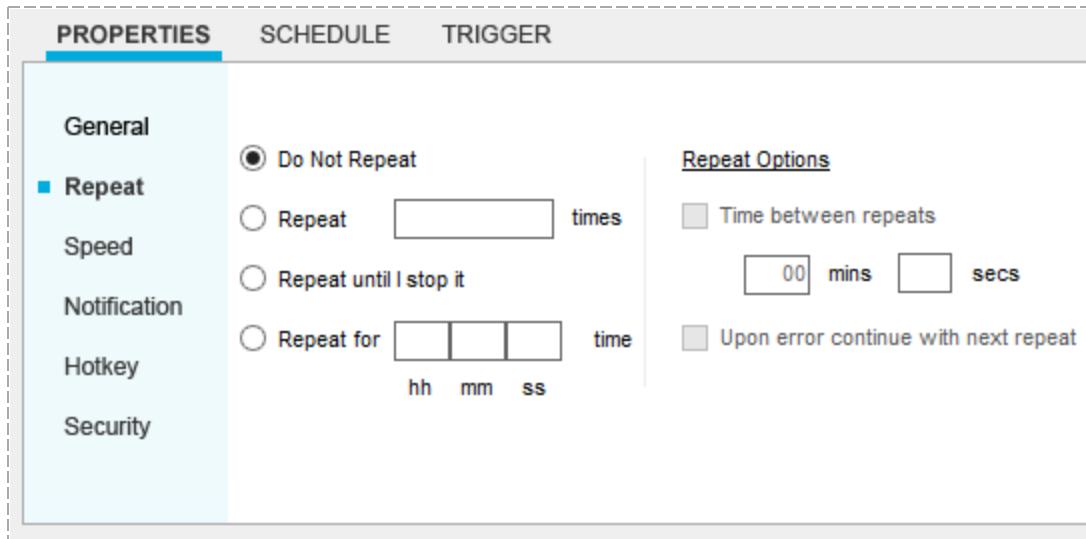
- Running a Task
- Scheduling Tasks to Run

Repeating a Task

Automation Anywhere enables you to repeat a task that is scheduled to run. This feature is especially useful while testing auto-mated tasks, or tasks that need to repeat upon run failure or even when running routine maintenance tasks.
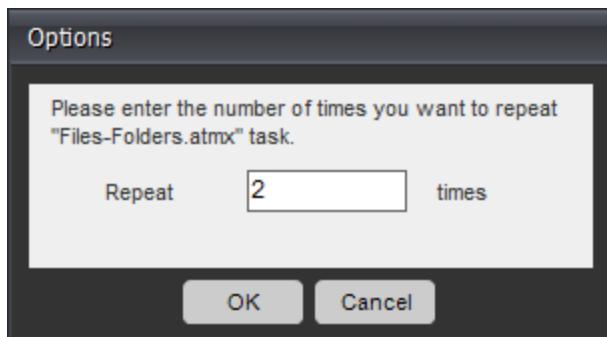
Use the Repeat property in the main Automation Anywhere window:
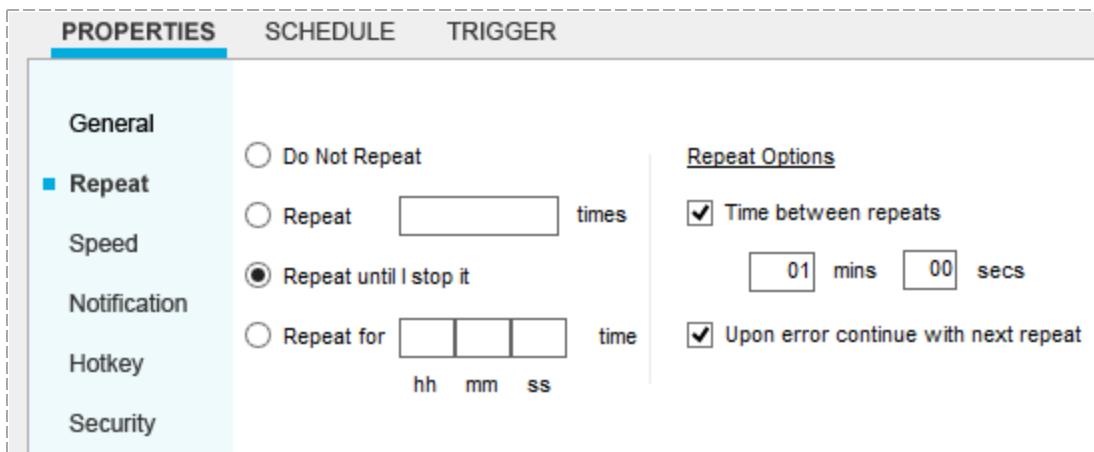
You can set the following options for the Repeat property:

- **Do not repeat:** The task runs only once. This is the default option.

- **Repeat a number of times:** Repeats the task a number of times, as specified in this field.
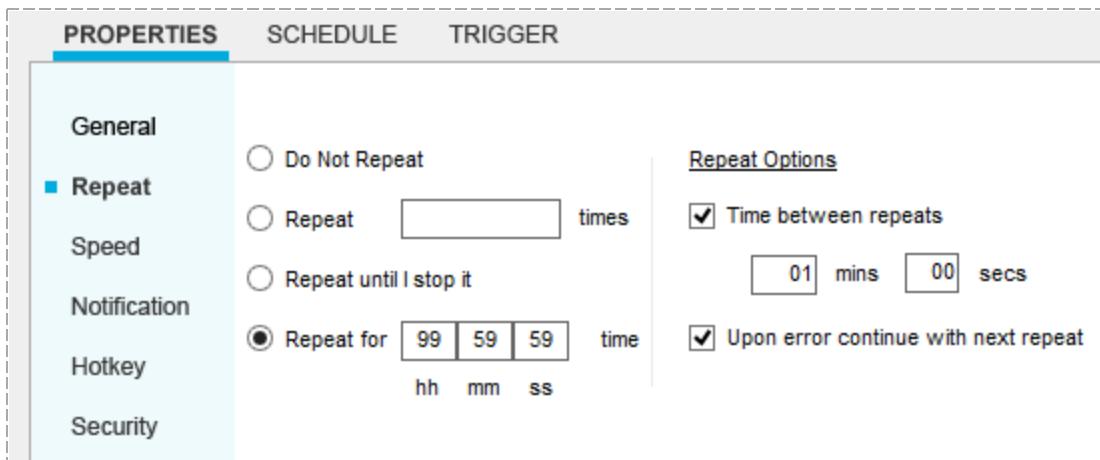
  Example: Repeat an email task 15 times to send 15 emails.

- **Repeat until I Stop it:** Repeats the task until you stop it by clicking the stop button or by pressing the ESC key. Use this option when you are not sure how many times the task needs to run.

- **Repeat for hh:mm:ss time:** Repeats the task for a time period that you specify.

> Note: You can specify up to 99 hours, 59 minutes, 59 seconds (a repeat duration of just over 4 days).
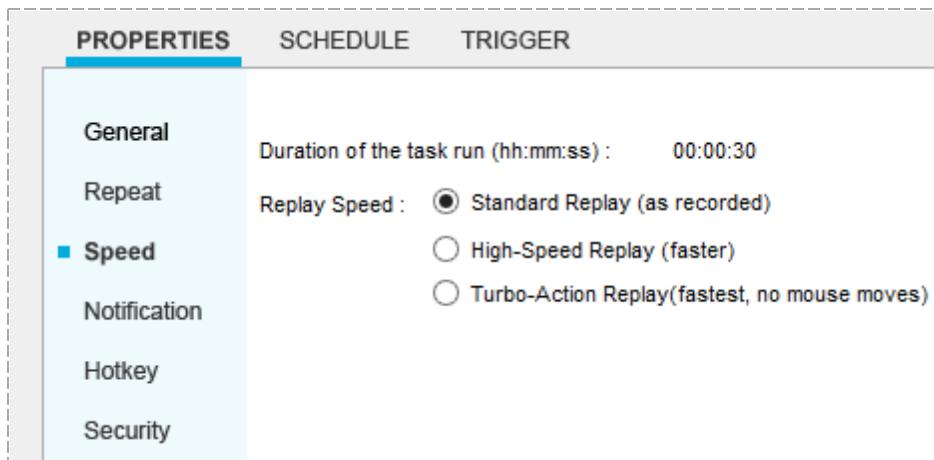
- **Time between repeats:** Sets a duration of time to wait before repeating a task.

- **Upon error, continue with next repeat:** By default, running tasks are stopped when an error occurs. By setting this option, the task is repeated regardless of run failure.

Also See...

- Viewing and Setting General Properties
- Adjusting the Speed of a Task
- Sending Email Notifications for a Task
- Setting up Hotkeys for a Task
- Setting Security Features for a Task

Adjusting the Speed of a Task

Use the Speed property to adjust the speed at which a task runs. You can also view the duration in hours, minutes, and seconds that it takes the task to run.

The speed of the task can be adjusted to run in 3 speeds:

- **Standard Replay** - The task runs at the same speed as when it was recorded.
- **High-Speed Replay** - The task runs at a faster speed than was recorded.
- **Turbo-Action Replay** - The task runs at the fastest speed and excludes any mouse moves.

The Automation Anywhere SMART Automation Technology ensures that when you increase the replay speed, accuracy and reliability are not compromised.

```
Tip: In some rare cases, certain pauses during the recording of a process have implicit applic-
ation-specific meaning. In these cases, set the replay speed to the Standard Replay mode to ensure
the most accurate replay.
```
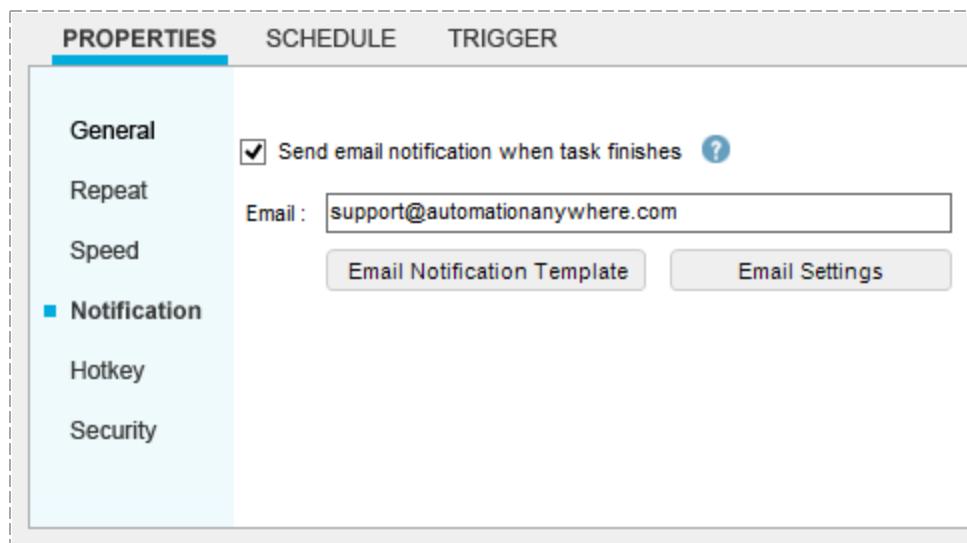
Also See...

- Setting Task Properties
- Scheduling Tasks to Run

Sending Email Notifications for a Task

When planning to run your tasks, you can set up email notifications that are sent to one or more email addresses that you specify.

In the Notification tab under Properties, select 'Send email notification when task finishes' and specify one or more email addresses.

If you select 'Send email notification when task finishes' check box, but do not specify an email address, Automation Anywhere uses the email address that is specified in the 'To:' field in the Tools -> Options -> Email Notification.

Tip: If you only need to specify one email address to notify, set it using the Tools menu and selecting Options -> Email Notification. However, if you need different tasks to send status emails to different email addresses, specify these email addresses for each task individually.Refer Setting up Email Notifications for details.
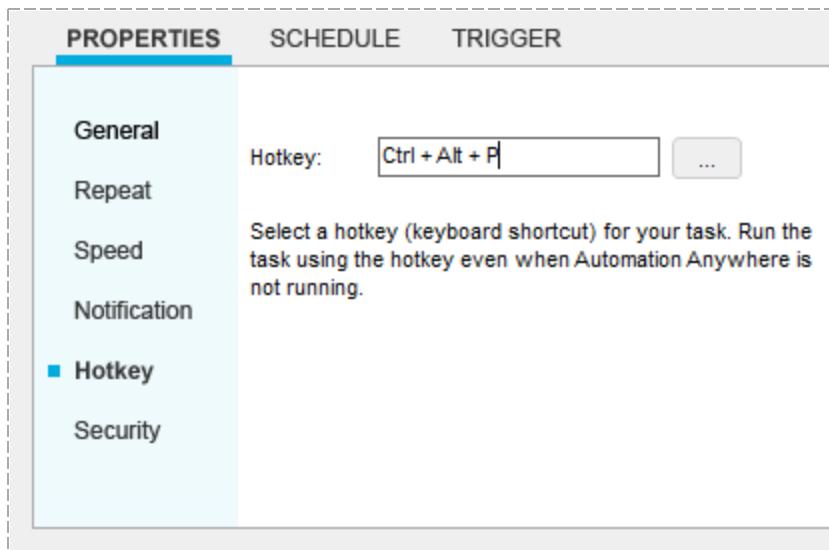
Also See...

- Setting Task Properties
- Editing a Task
- Deploying Tasks to Run Remotely

Setting up Hotkeys for a Task
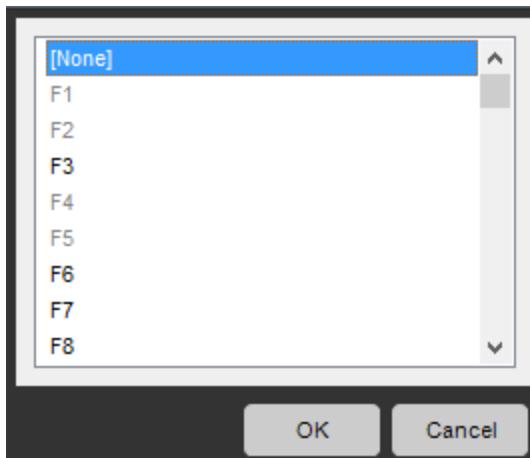
One of the distinguishing benefits of automating your tasks with Automation Anywhere is the ability to launch a task with the press of a single key.

You can assign a hotkey to an automated task by using the Hotkey property.

To assign a hotkey, follow these steps:

1. On the Properties tab, select Hotkey.
2. Either specify a hotkey in the field provided or select a hotkey from the pop-up window. If a hotkey is being used by another process, it will be show as not selectable in the list.

> Note: The automated tasks can run even when the Automation Anywhere client is not running.

Also See...

- Stopping a Task
- Scheduling Tasks to Run

Setting Security Features for a Task

Automation Anywhere provides advanced security for running tasks in a secure manner through password-protection. Password-protected tasks are encrypted with strong, industry-standard encryption.

The Security* property provides the following features:

1. **Prompt and Verify Password:** When you specify a password, you can specify whether you want Automation Anywhere to prompt the user for the password before editing the task and before running the task. Simply check the options that you require for your security.

2. **Auto-Login Advanced Settings:** You can specify the following advanced settings for Auto-Login:

- **Running this task in stealth mode:** Use this option to prevent information from being displayed on the screen while the task is running in Auto-Login mode. Application windows and programs are hidden from your screen to avoid unauthorized users seeing the information.

- **Disable mouse and keyboard for this task:** Use this option to disable the computer mouse and keyboard while the task is running in Auto-Login mode. Unauthorized users are prohibited from gaining control of the computer while the task runs. If the computer recognizes an unauthorized user who is accessing the computer, Automation Anywhere automatically locks the computer, ensuring security.

```
Note: When nesting automation tasks, if a parent task runs a child task, the Auto-Login
settings of the parent take effect, regardless of the settings of the child.
```
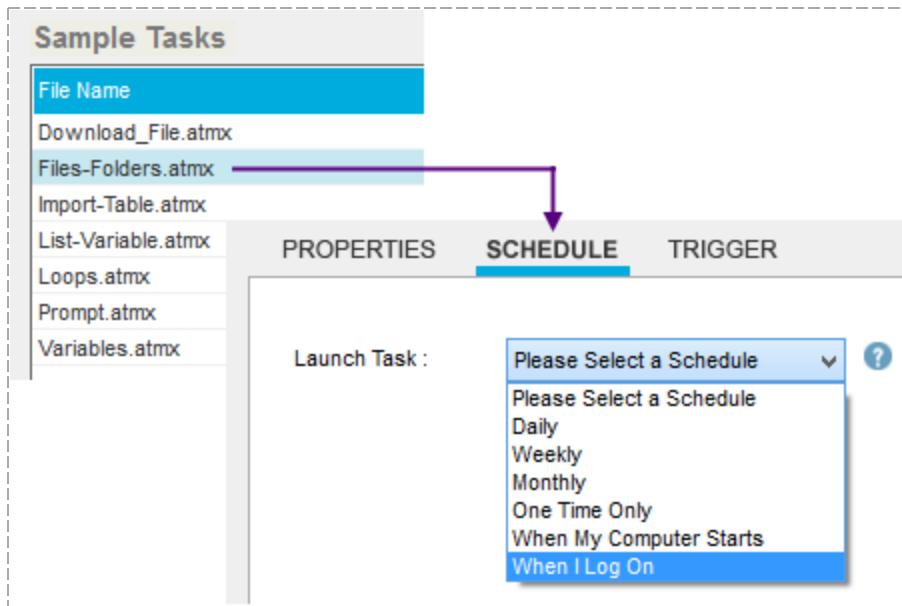
The security feature is not available in the Trial version.

Also See...

- Running a Task
- Scheduling a Task

Scheduling Tasks to Run

After creating an automated task, you can schedule the tasks to run based on a schedule or on a trigger event.

Scheduling Tasks

Automation Anywhere provides a Scheduler and a Schedule Manager that you can use to run your tasks anytime you want.

Scheduling Tasks Using the Scheduler

The Automation Anywhere Scheduler provides time-based scheduling capabilities. You can schedule tasks to run on particular days, weeks, or months of the year.

For more information on the Scheduler, see Scheduling Tasks Using the Scheduler.

Viewing Task Schedules Using the Schedule Manager

The Schedule Manager enables you to view, add, or edit the schedules of tasks that you have created.

For more information on the Schedule Manager, see Using the Schedule Manager.

Scheduling Tasks Using Triggers

To run a task based on a trigger event, use the Trigger tab on main Automation Anywhere window.

Some examples of events that can trigger a task to run include when:

- A new window opens
- A specific file is created
- A file is deleted
- An email is received in a particular in-box

For more information about using triggers, see Using the Trigger Manager

Also See...

- Recording an Automation Task
- Starting the Automation Anywhere Client
- Installing the Automation Anywhere Client

- Using Special Keys
- Setting Task Properties
- Exiting the Automation Anywhere Client
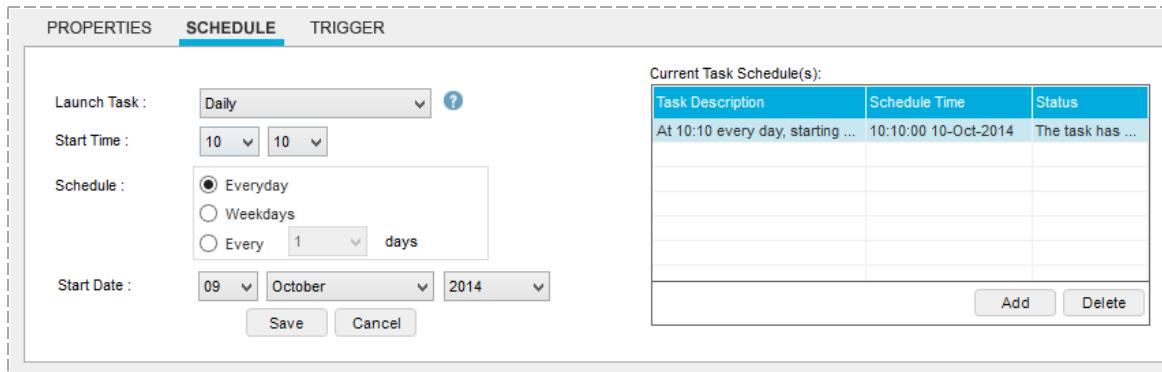
Scheduling Tasks Using the Scheduler

The Automation Anywhere Scheduler enables you to schedule automated tasks to run at preferred times, without requiring supervision or intervention.
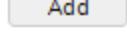
Scheduling Tasks

Automated tasks and process workflows can be scheduled to run as shown:

Schedule a task using the Scheduler by following these steps:

1. Select the task that you want to schedule.
2. Click the Schedule tab.
3. Set the scheduling information for time, date, and frequency.
4. Click on Save.
5. Provide your Windows user name and password.
6. Schedule a task to run, multiple times. To add more scheduled times, click the Add [Add] button.

To ensure the computer is unlocked when a scheduled task runs, see Using the Auto-Login Option.

Automation Anywhere provides scheduling options for running tasks with the following frequencies:

- Daily (for example, every day, only weekdays, or every 2 days)
- Weekly (specific days of the week; for example, Mondays)
- Monthly (specific days of the month; for example, January 1 and October 5)
- One time only
- When the computer is started
- When you log onto the computer
- When the computer is idle

> Note: Schedules not visible after upgrade? Reference 'Migrating schedules to newer versions'.

Entering Windows Login Information

For security reasons, when scheduling tasks to run using the Scheduler, you may be required to enter the Windows user name and password. This security check ensures that only the machine owner can schedule tasks to run.

No User Name and Password?

Many automation users do not have passwords set on their systems. In this case, you can create a password to enable you to schedule tasks. Use the Auto-Login option to avoid needing to enter the password each time you start the computer.

To create a password on your system, follow these steps:

For Windows XP

1. On the Windows XP desktop, click the Start button and click on the Control Panel.
2. Double-click on User Accounts.
3. Click on the user account to which you want to add a password, and click Create a Password.
4. Complete the instructions for creating a password. Any valid password is acceptable, although it cannot be a blank password.

To configure your machine to use the Auto-Login option, which logs you in each time it restarts, follow these steps:

1. On the Windows XP desktop, click the Start button and click Run.
2. Type: control userpasswords2
3. Un-check the check box for 'Users must enter a username and password to use this computer' and click Apply.
4. Enter the user name and password to automatically log onto the system, and click OK.
5. Click OK again.

Note: You must log off and log on again before scheduling a task.

For Windows 7

1. On the Windows 7 desktop, click the Start button and click on the Control Panel.
2. Double-click on User Accounts.
3. Complete the instructions for creating a password. Any valid password is acceptable, although it cannot be a blank password.

To configure your machine to use the Auto-Login option, follow these steps:

1. On the Windows 7 desktop, click the Start button and click Run.
2. Type: control userpasswords2. The User Accounts window is displayed.
3. Go to the Advanced tab
4. Deselect the 'Require users to press Ctrl + Alt + Delete

Note: Enabling Auto-Login makes the computer more convenient to use, but can pose a security risk. Be aware that any person can turn on the machine and access your files.

Also See...

- Scheduling Tasks to Run
- Scheduling Tasks Using Triggers
- Setting Task Properties
- Setting Security Features for a Task

Using the Schedule Manager

The Automation Anywhere Schedule Manager is a convenient facility that displays all task schedules that are set for your tasks. Using the Schedule Manager, you can view, add, or edit any of your task schedules.

Viewing Task Schedules

To view your task schedules, launch the Schedule Manager in one of the following ways:

- In the main Automation Anywhere window, click on the Schedule Manager tab on the lower left side.
- Or click on Tools -> Schedule Manager on the menu bar.
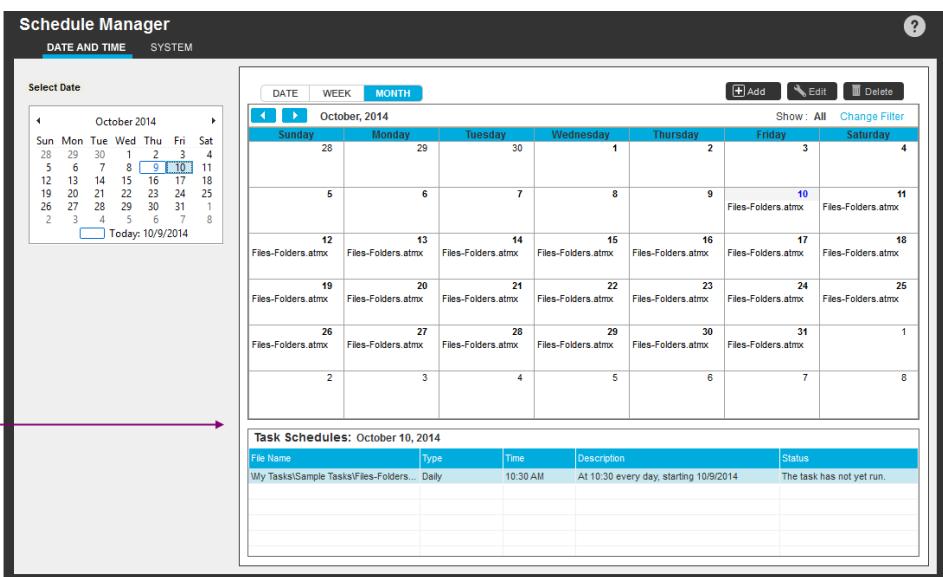
> Note: Schedules not visible after upgrade? Reference 'Migrating schedules to newer versions'.

Creating, Editing, and Deleting Task Schedules

You can manage all of your task schedules by using the two tabs in Schedule Manager: **Date and Time** tab and **System** tab.

Date and Time Schedules

Within the **Date and Time** tab, you can use the Day, Week, and Month tabs to work with scheduled tasks.
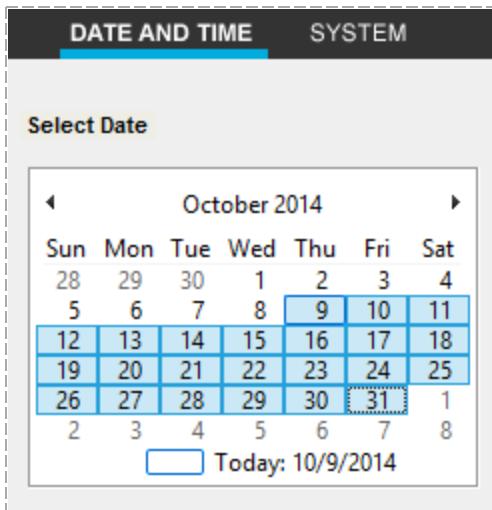
You can add, edit, or delete scheduled task to run by using any of these views:

- **Day:** The Day view displays the time and description for each scheduled task. You can add, edit, or delete scheduled tasks. Use this if you need to run your tasks daily and with more frequency.
- **Week:** The Week view displays the date and time of scheduled tasks for the week, and includes a description. Use this option when you wish to schedule to run your task on a weekly basis.
- **Month:** The Month view displays the date and description of scheduled tasks for the month. Use this option when the frequency of running your tasks could be scheduled monthly.
- **Date Range:** In the Select Date section on the left, you can specify a date range for scheduled tasks by clicking and holding the mouse button on a start date and dragging it to the end date, highlighting the range. The Month view automatically opens

to display scheduled tasks within that date range.

System Command Schedules

Use the **System Tab** to view tasks that are scheduled to run based on system events, such as triggers.

Using this view, you can add, edit, or delete task schedules that use triggers to initiate a run.
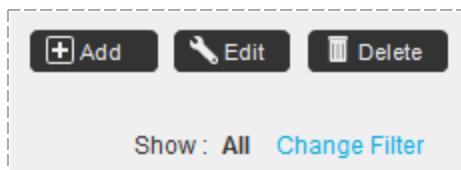
The following system triggers are available:

- When My Computer Is Idle*
- When My Computer Starts
- When I Log On

Applicable to Windows XP only
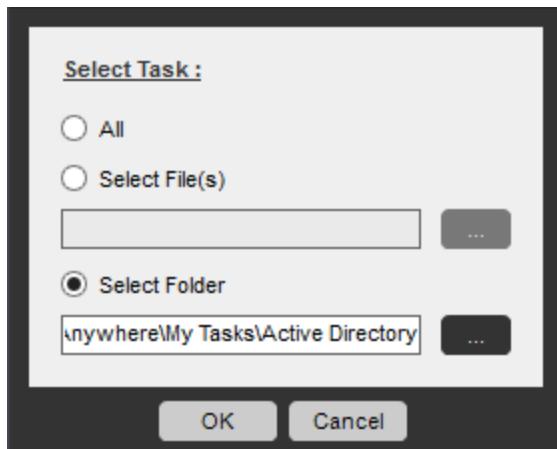
Changing the Filter View of Task Schedules

You can change the view of scheduled tasks by clicking on Change Filter on the extreme right of the systems tab in the Schedule Manager.

Select tasks to view by taking one of the following actions:

- Browse to the required file or folder, and select the task.
- Select multiple tasks by holding the Control (Ctrl) key and selecting the tasks.
- Select "All" to view all scheduled tasks for the system.

Also See...
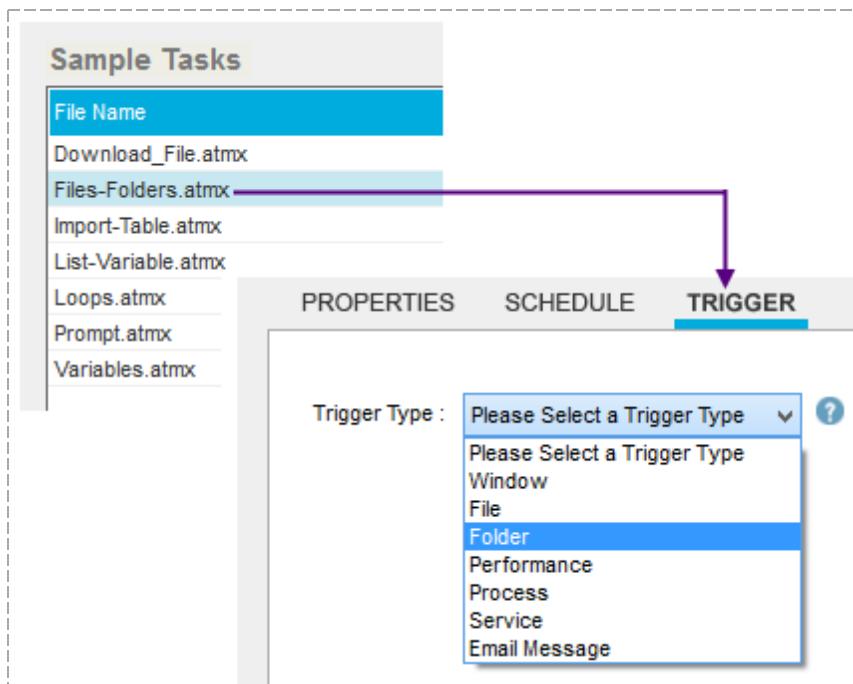
- Scheduling Tasks to Run
- Using Triggers

Scheduling Tasks Using Triggers

In addition to scheduling a task to run based on a date and time, you can use trigger events to start your automated tasks.

The Trigger feature enables a task to run automatically in response to an event that occurs on your computer. You can use triggers to run a task when, for example, a new window opens or a specific file is created.

To use triggers, select an automation task and click on the Trigger tab in the main Automation Anywhere window and select the trigger type that suits the selected task.
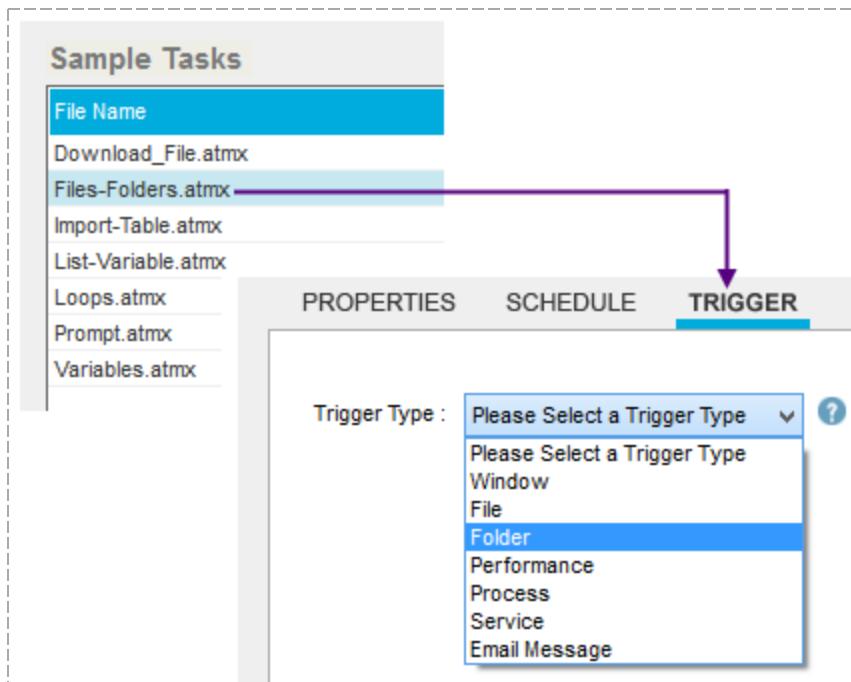
Also See...

- Adding a Trigger to a Task
- Managing Triggers Using the Trigger Manager

Adding Triggers to a Task
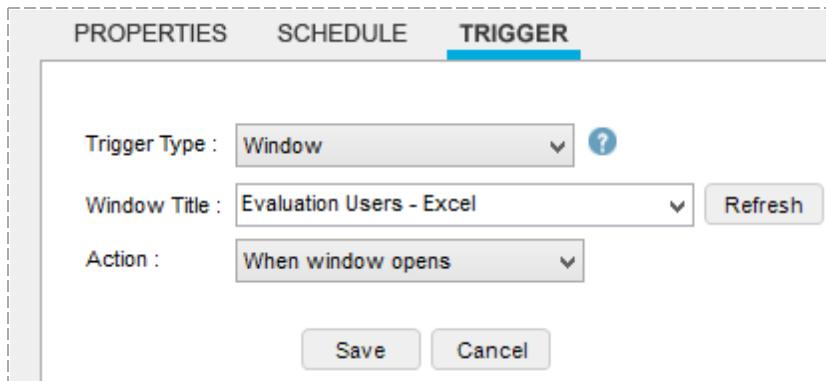
An Automation Anywhere *Trigger* automatically runs a task in response to an event that occurs on your computer. You can use triggers to run a task when, for example, a new window opens or a specific file is created.

Automation Anywhere provides the following types of trigger events:

Using Window Triggers

Window triggers launch an automated task when an application window action takes place. Automation Anywhere provides the list of open applications on your computer and enables you to refresh the list at any time. You can select from either of the following options as the trigger event:

- When an application window opens
- When an application window closes

Using File Triggers
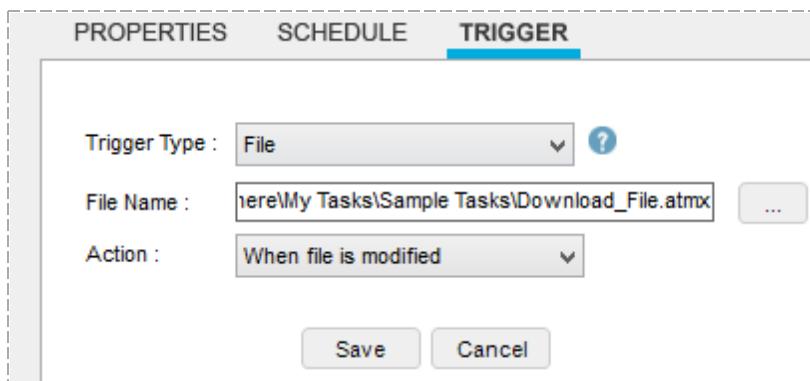
File triggers launch an automated task when a file action takes place. You can select from any of the following options as the trigger event:

- When a new file is created
- When an existing file is deleted
- When a file is renamed
- When a file is modified

Using Folder Triggers
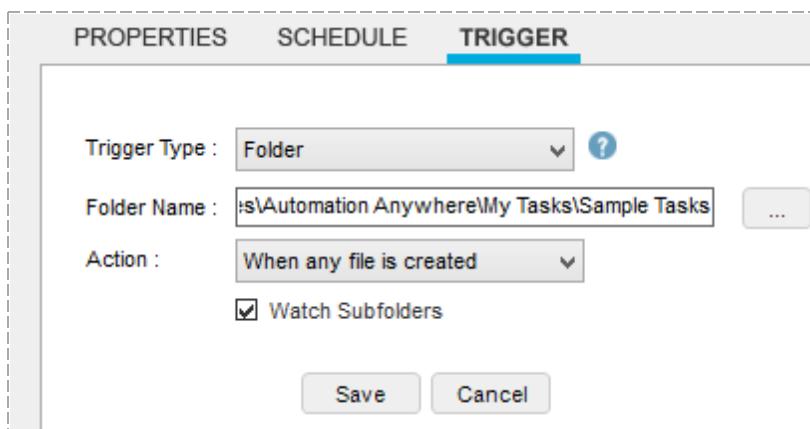
Folder triggers launch an automated task when events affect folders on your computer. You can select from any of the following options as the trigger event:

- When a new file is created in the folder
- When an existing file is deleted in the folder
- When a file is renamed in the folder
- When a new folder is created
- When an existing folder is deleted
- When a folder is renamed
- When a folder is modified

Using Performance Triggers

Performance triggers launch an automated task when particular computer resources reach a threshold or certain criteria are met. You can select from the following options as the trigger event:

- **CPU Usage:** The computer CPU usage reaches a level that you specify (in %).
- **Free Disk Space:** A certain amount of free disk space is reached that you specify in megabytes (MB).
- **Process Count:** When a particular number of processes that you specify are running.

Using Process Triggers

Process triggers launch an automated task when a system process starts or stops. Automation Anywhere provides the list of available processes for your computer and enables you to refresh the list at any time. You can select from either of the following options as the trigger event:

- When a process starts running
- When a process stops running

Using Service Triggers
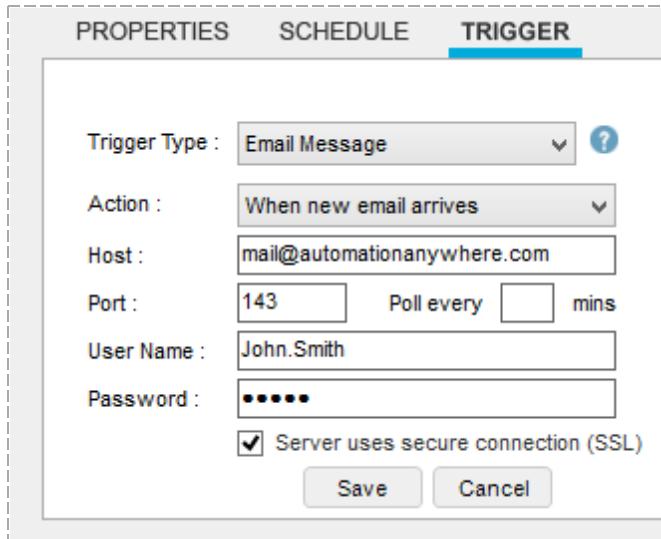
Service triggers launch an automated task when a system service takes action. Automation Anywhere provides the list of available services for your computer and enables you to refresh the list at any time. You can select from any of the following options as the trigger event:

- When a service starts running
- When a service stops running
- When a service resumes running
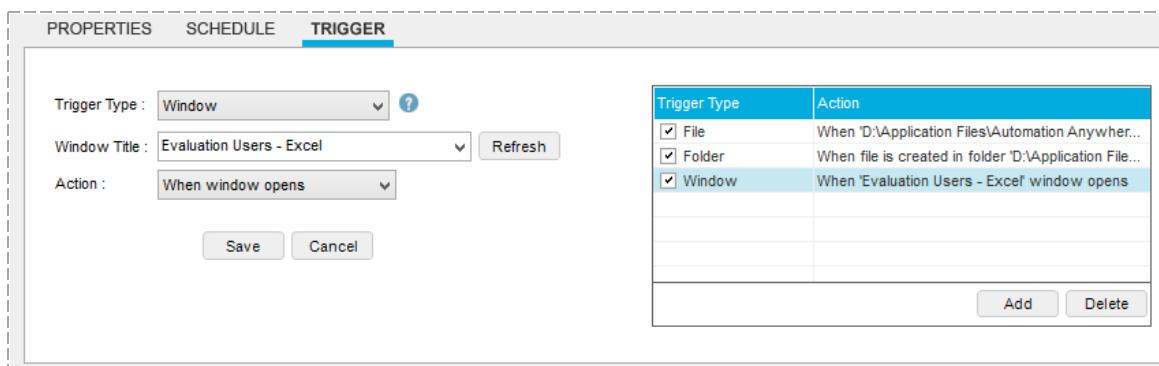- When a service pauses

Using Email Message Triggers

Email Message triggers launch an automated task when an email is received in the email account that you specify. To specify a trigger for new incoming email messages on a mail server, specify the following:

- Host name
- Port details
- User name
- Password
- A time interval to use between checking for new messages
- If the email account uses SSL, check the "Server Uses Secure Connection (SSL)" check box.

Examples of Using Triggers

Example 1: Run an automated task each time a 'Microsoft Excel' window opens:

1. For Trigger type, select Window.
2. From the Window Title drop-down menu, select the title *Evaluation Users - Excel*. If the window is not available in the list, open the application and click on Refresh.
3. Under Action, select 'When window opens'.
4. Save the trigger by clicking the Save button. The task will now run each time Microsoft Excel opens.

Example 2: Run an automated task that sends an email whenever a file is deleted from a critical work folder on the computer.

1. For Trigger type, select Folder.

2. For Folder Name, click on the ellipsis (...) and select the folder that you want to monitor.

3. For Action, select 'When any file is deleted.'

4. Save the trigger by clicking on the Save button. When a file is deleted from the selected folder, the task will run and send a notification email.

Recommendations for Using Triggers

Automation Anywhere recommends you follow these guidelines with using triggers to run your tasks:

- If you are running a task on a locked computer, use the Auto-login feature to unlock the computer when the triggered task runs. See Using the Auto-Login Option. Refer Using the Auto-Login Option for details.

- You can assign a system variable to a trigger action that returns the name of the file, folder, or window that used to trigger the task. Refer Trigger Variables for details

- Use triggers to run process work flows. Refer Scheduling Tasks Using Triggers for details.

Also See...

- Using the Trigger Manager

Using the Trigger Manager

Over time, you might create several triggers that run various automated tasks. For example, a single task might be run using five different triggers, or a single trigger might run five different tasks. As the number of tasks and triggers grow in number, you will want to organize and manage these triggers.
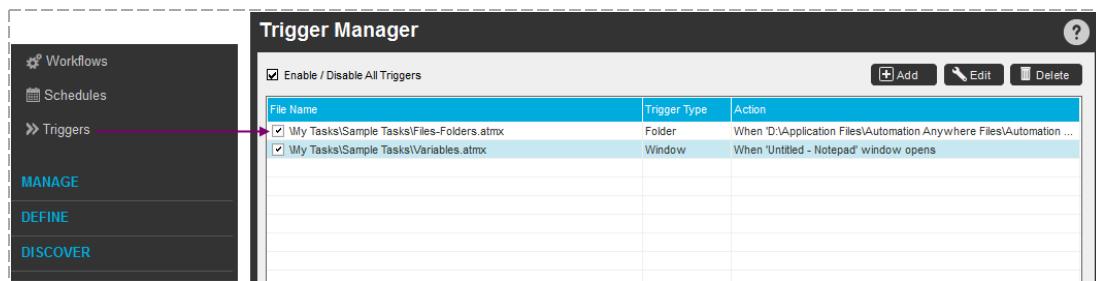
You can use the Trigger Manager to display, add, modify, and delete triggers you have set for your tasks.

Using the Trigger Manager, you can:

- Add a trigger
- Modify a trigger
- Delete a trigger
- Enable a trigger
- Disable a trigger
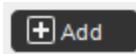
To use the Trigger Manager, follow these steps:

1. In the main Automation Anywhere window, click on Trigger Manager tab on the left side. The Trigger Manager window is displayed, showing the tasks that are driven by each trigger.

2. Add additional triggers or work with your existing triggers:

- To add a new trigger, click the Add button.

- To edit an existing trigger, click the Edit button.

- To delete an existing trigger, click the Delete button and click "Yes" to confirm.

- To enable a trigger, ensure that the check box in the far left column is checked.
- To disable a trigger, un-check the check box in the far left column.

You can enable or disable all triggers at once by checking or un-checking the check box at the top of the Trigger Manager window.

Also See...

- Scheduling Tasks Using Triggers
- Scheduling Tasks to Run

Troubleshooting Scheduled Tasks

Having trouble running a scheduled task?

If your scheduled task fails to run, follow these steps:

1. **Ensure that you have provided the user name/password.**

   For security reasons, you may be required to enter your user name and password to run a scheduled task. For more inform-ation, see " For Windows XP only: Entering Windows Login Information."

2. **If you have multiple accounts, verify that the correct user name/password are used.**

   If you have multiple accounts on the same computer, such as a domain account and a local account, verify that the user name you specified is the same user name that was used to log into the computer. Scheduled tasks will not run if the sched-uled task attempts to run under a different user account. Automation Anywhere pre-fills the user name field with the user name you used to log in.

3. **Check the task status in Control Panel.**

   To check the status of the task in the Control Panel, follow these steps:

   1. Open the Control Panel and click on **Scheduled Tasks**.
   2. Select the task that corresponds to your task (the file starts with your task name).
   3. Right-click on the task and select Run. If the task does not run, check the Status column.
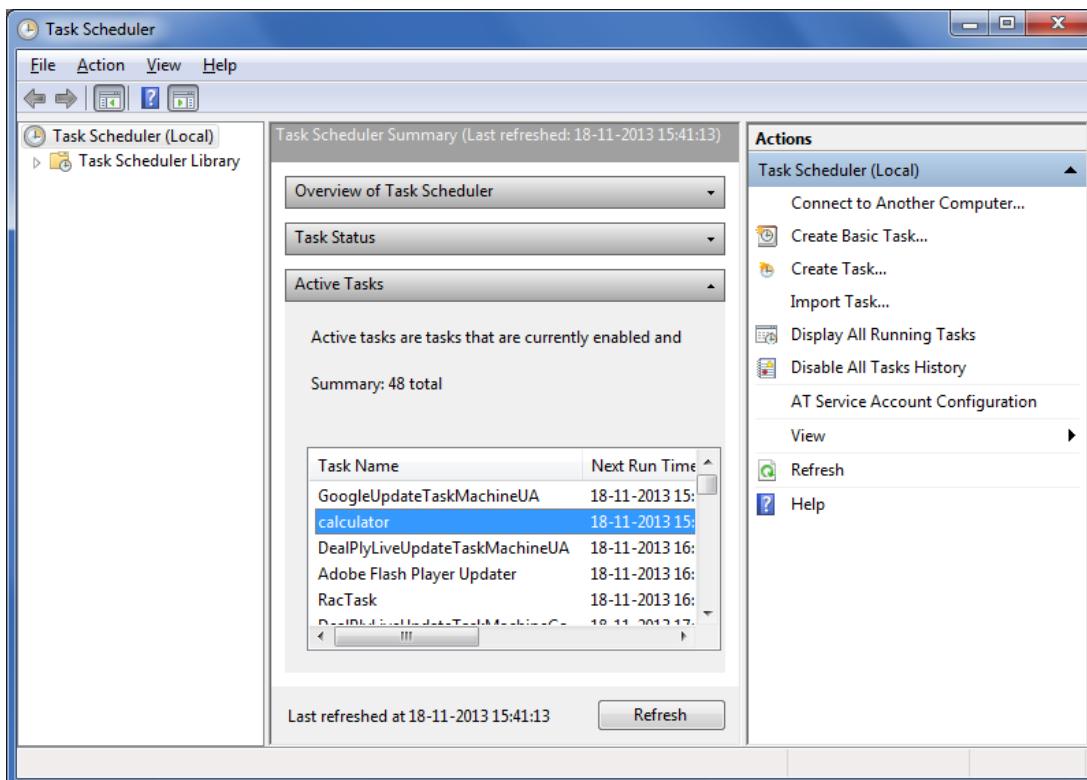
4. **Schedule the Calculator Application.**

   If your task still does not run, follow these steps:

   1. Open the Control Panel and click on **Administrative Tools**.
   2. Click on the Task Scheduler.
   3. In the Active Tasks drop-down, select Calculator (see figure below). If the calculator does not run, the scheduler service is not installed properly on your computer.

5. **Verify that the Task Scheduler service is started.**

By default, the Scheduler service is started on all Windows XP machines. Verify that the service is started by following these steps:

1. Open the Control Panel and click on Administrative Tools. Select Services.

2. Verify that the status of the Task Scheduler service is "Started."

3. If the status is not "Started", right-click on it and select Start.

4. Verify that the 'Startup type' is set to 'Automatic'. This ensures that the service will start automatically in the future.

6. **Restart the Task Scheduler.**

   If the problem persists, restart the Task Scheduler by following these steps:

   1. Open the Control Panel and click on Scheduled Tasks.

   2. In the Advanced Menu, click on 'Stop using Task Scheduler.'

   3. Wait 10 minutes.

   4. Click on "Start using Task Scheduler" to restart the Task Scheduler service.

7. **Contact Technical Support.**

   After completing these steps, if you are not able to run the task as scheduled, it is likely that the scheduler service is not installed correctly on the computer. Contact Technical Support at: Open a Support Case.

Also See...

- Scheduling Tasks to Run
- Running a Task
- Debugging Tasks
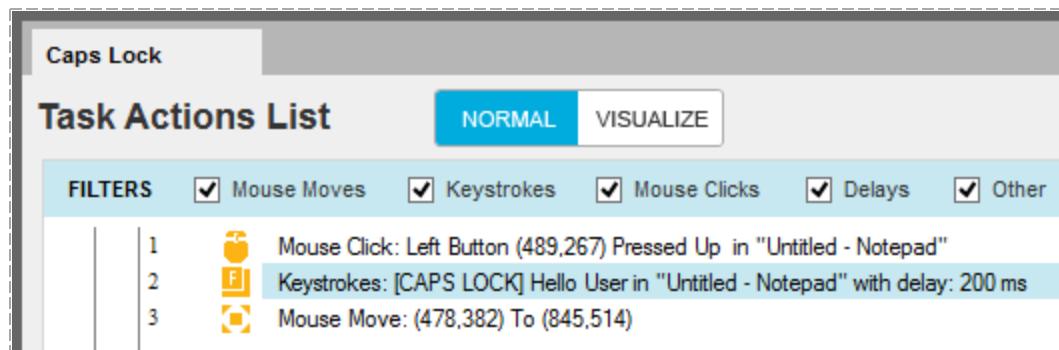
Using Caps Lock in Your Tasks

As a best practice, make sure that the **Caps Lock** key is turned OFF prior to creating your automation tasks.

When you create a new automation task, Automation Anywhere records the state of the Caps Lock key during the recording.

When running the task, Automation Anywhere changes the state of Caps Lock key to match the state that existed prior to recording the task.

Recording with Caps Lock ON

If the Caps Lock key is turned ON when you start recording, Automation Anywhere records the task with Caps Lock ON.

When you run this task:

- Automation Anywhere checks the state of the Caps Lock key.
- If the Caps Lock key is OFF at the time it is run, Automation Anywhere turns Caps Lock to ON before running any of the steps.
- If you use Insert Keystrokes commands within the task, upon running the task Automation Anywhere toggles the state of Caps Lock to OFF, resulting in lowercase characters.

Recording with Caps Lock OFF

If the Caps Lock key is turned OFF when you start recording, and you use Insert Keystrokes commands within the task to type uppercase characters, Automation Anywhere turns Caps Lock ON.

> Note: Be aware that when you use Insert Keystrokes commands, no indicator exists for changes to the Caps Lock key status. Changes are carried out internally.

Also See...

- Debugging Tasks

Deploying Tasks to Run Remotely

Some situations are ideal for deploying automated tasks to remote computers so they can be run by users without requiring the users to be familiar with the technology. Automation Anywhere enables you to convert your automated tasks to standalone programs (.EXE files).

An .EXE file can run on any computer without the need for that computer to be running Automation Anywhere. The .EXE files are intelligent files that contain a SMART Automation agent that is embedded within the .EXE file. The SMART Automation agent provides the same intelligent automation and auto-sensing technology provided by Automation Anywhere.

When you run an .EXE file on other computers, the SMART automation technology adjusts for over 40 possible scenarios, such as window size, position changes, CPU load, desktop icon location changes, and more. The .EXE also replaces system variables, such as dates and user-defined variables, to ensure smooth task compatibility across computers.

With the Task-to-EXE capability, businesses and IT departments can manage and configure software on numerous machines, alter software behavior, create interactive scripts that accept user input, and much more.

Licensing Task-to-EXE Capability

Automation Anywhere provides licensing options for creating executable tasks. Depending on the number of machines on which you want to run the tasks, the pricing can vary. Be sure to contact your account representative if you have questions about running tasks remotely.

Software Requirements

To remotely run tasks using Task-to-EXE, the following .NET Framework(s) must be installed on the machines for which .EXE files will run:

•.NET 3.5 + SP1

•.NET 4.0

```
Note: If using Enterprise Task-to-Exe, you need to have both versions of .NET Framework
installed; for Small Business, Premier and Standard Task-to-Exe, only .NET Framework 4.0 should suf-
fice.
```

Also See...
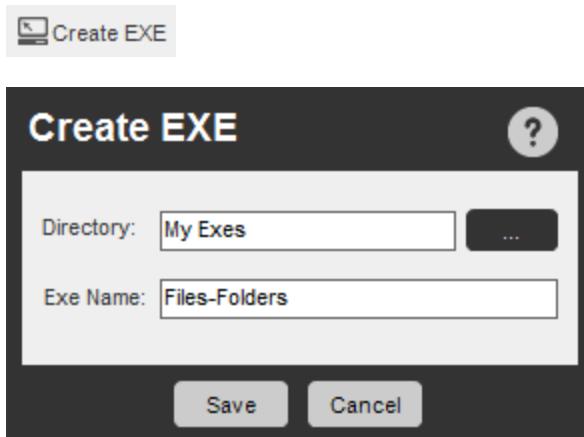
- Converting a Task to a Standalone EXE
- Examples of Task-to-EXE Conversion

Converting a Task to a Standalone EXE

Available with Create Exe license in Small Business and Enterprise editions only

To convert an Automation Anywhere task to an .EXE file, follow these steps:

1. Select the task from the Task List.
2. Click Create .EXE button to invoke the create exe window

3. Browse and select the directory for the .EXE file.
4. Type a name for your .EXE file and Click Save.

💡 Tip:
  1. To improve accuracy while running .EXE files across multiple machines, use keyboard shortcuts and keystrokes instead of mouse clicks while recording tasks.
  2. To improve task efficiencies, use the Web recorder to capture Web controls and the Object Recorder to capture Windows controls while recording tasks.

Also See...

- Examples of Task-to-EXE Conversion

Examples of Task-to-EXE Conversion

The following example illustrates how useful the Task-to-EXE conversion capability can be to an organization.

Use Case Scenario

Automate Installation

Create a task that automates the installation of an application:

1. Create a task that installs the application on the main machine.
2. To deploy the application to other machines, create an executable (.EXE) of the task using the Create EXE option.
3. Save the .EXE file in `My Documents\Automation Anywhere Files\Automation Anywhere Files\My Exes.`
4. Copy and paste the .EXE to the local machines on which you want the application to be installed.

Debugging Tasks

For advanced users, Automation Anywhere provides the capability to debug automation tasks. Although Automation Anywhere enables you to easily create simple tasks, some longer, more complex tasks might require debugging.

Automation Anywhere provides a Debug Toolbar to manage your debugging. Follow these steps to enable debugging mode:
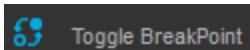
1. Display the Debug Toolbar either by clicking on the Enable Debugging button or by clicking on the Debug menu and selecting **Enable Debugging**.
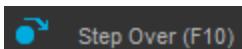
   The ' Variable(s) Watch Table' window is displayed. To understand more about this feature refer Watching Variables.
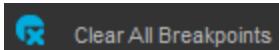
2. Use the Debug Toolbar to insert Breakpoints in your task. To insert a Breakpoint, select a command and insert a Breakpoint by either clicking the Toggle Breakpoint button on the Debug Toolbar or by pressing the F9 function key.
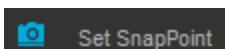
3. Debug the task action-by-action by using the Step Over button (or the F10 function key).
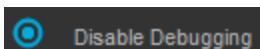
4. Run your task in Debug mode (one command at a time) to isolate any errors.

5. To remove a Breakpoint, select the command and either click the Toggle Breakpoint button or press F9. To clear all Breakpoints, click the Clear All Breakpoints button on the Debug Toolbar.

6. Use the Set SnapPoint button in a task to capture images of the task while it runs. Use the Visualize button to view the images that are captured when using the Set SnapPoint option in Debug mode.

7. To stop debugging, either click on the Disable Debugging button or click on the Debug menu and select **Disable Debugging**.

> Note: Automation Anywhere does not store debugging information (Breakpoints); therefore, when closing the Task Editor window, be aware that all Breakpoint information will be lost.

Also See...

- Creating an Automation Task

Other User
Resources:
User forums
Ask the Expert
Open a support case
Automation Examples

Calling Tasks from Programs or Scripts

You can call Automation Anywhere tasks from other programs, scripts, or batch files.

By default, Automation Anywhere saves all tasks in the folder named:

```
My Documents\Automation Anywhere Files\Automation Anywhere
```

Calling a Task from a Batch File

Task-name: download-emails.atmx

- Call the task named `download-emails.atmx` from a batch file by adding the following line in the batch file:

```
"C:\Program Files\Automation Anywhere Enterprise 7.0\Client\AA.Player.exe"
```

```
"/fD:\My Documents\Automation Anywhere Files\Automation Anywhere\My Tasks\download-emails.atmx/e"
```

Calling a Task from an Excel Macro

Task-name: download-data.atmx

- To call the `download-data.atmx` task from an Excel macro, add the following text to the Excel macro:

```
Dim RetVal
```

```
RetVal = Shell("C:\Program Files\Automation Anywhere Enterprise 7.0\Client\AA.Player.exe "/fD:\My
Documents\Automation Anywhere Files\Automation
```

```
Anywhere\My Tasks\download-data.atmx"/e", 1)
```
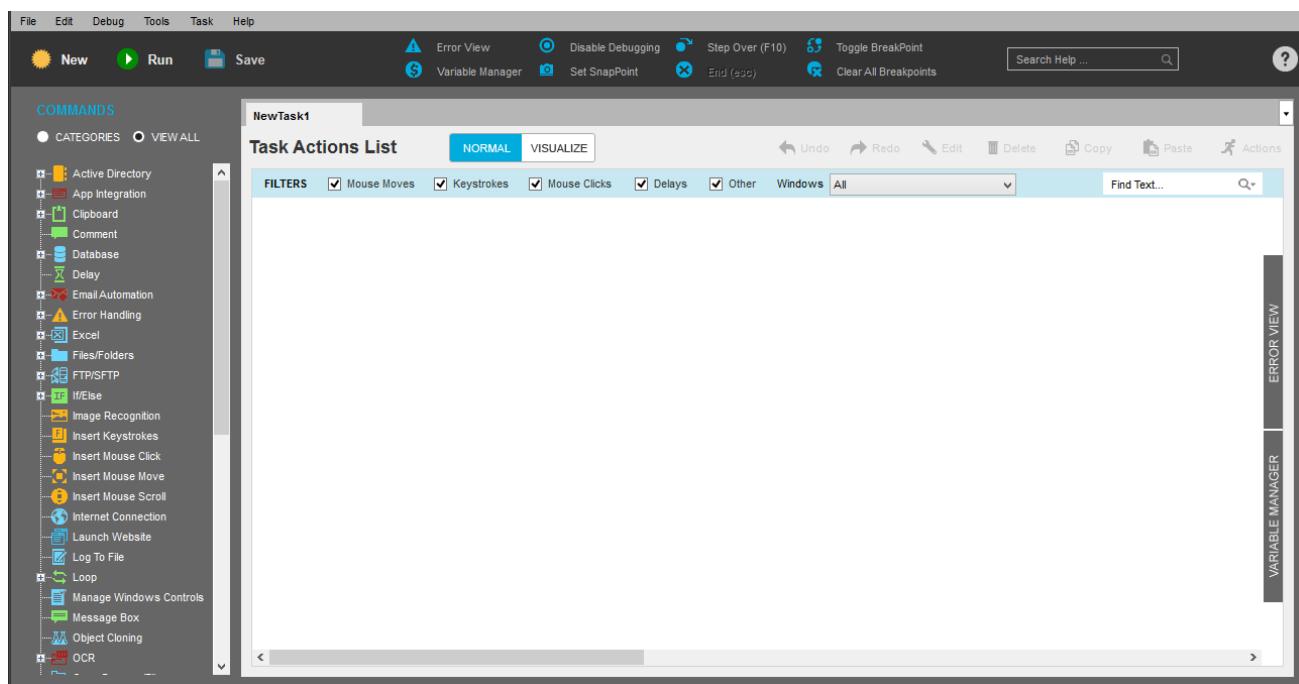
- If the task is successful, it returns 0 to the calling program. If it encounters an error, it returns 1.

Using the Task Editor

The Automation Anywhere Task Editor is the most powerful component of the product, providing business process specialists and IT staff with all the tools necessary for enhancing, streamlining, and deploying automated processes.

The Task Editor provides hundreds of commands that you can use to create automated processes for your organization. It is unique in the software automation industry: non-technical business users can quickly build and deploy automated processes with simple drag-and-drop operations, yet the solution is flexible and robust enough for technical IT staff to build more complex, integrated, end-to-end processes.

(Image as seen in Enterprise Edition 8.2)

Opening the Task Editor

From the main window, you can open a task in the Task Editor the following ways:

- Select a task from the Task List and click the Edit button.

- Double-click on a task in the Task List, provided that double-click editing is enabled in the Advanced Settings for Edit Task. By default, this setting runs a task when double-clicking. For more information, see Editing a Task.

- Create a new task in the Task Editor by clicking on the New button, and select Task Editor.
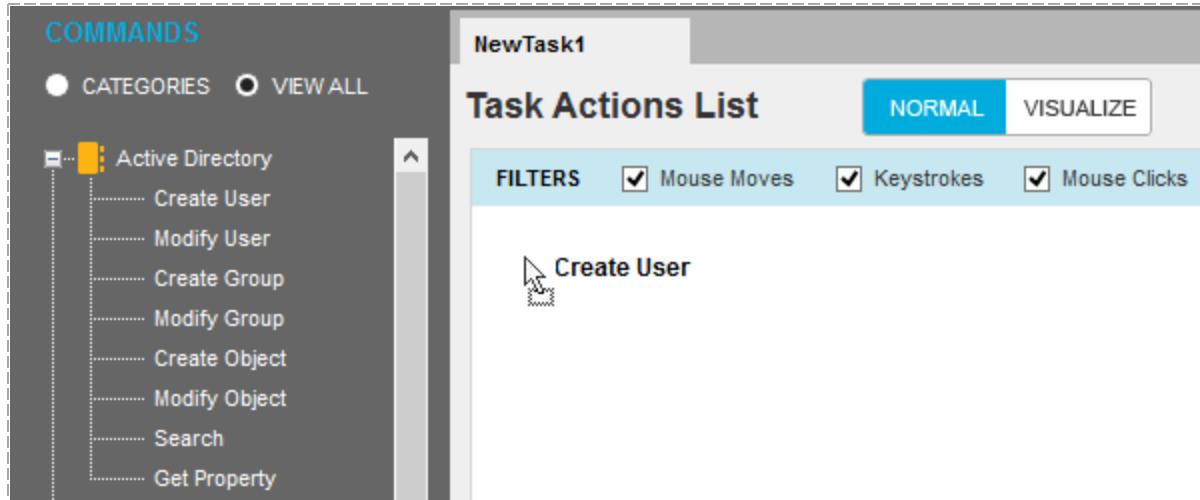
Task Editor Components

The Task Editor provides the following components that make up the user interface:

- Commands panel
- Task Actions List
- Error View and Variable Manager panel
- Action buttons
- Filters

Using the Commands Panel
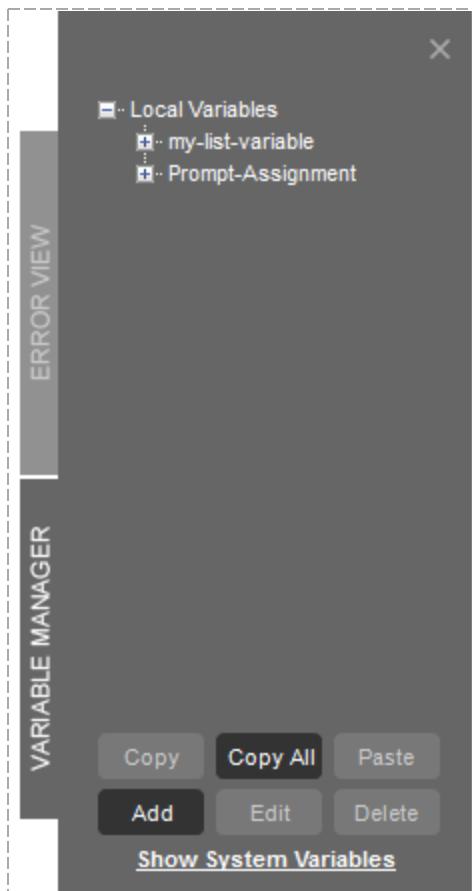
The left panel contains all of the Automation Anywhere commands that you can use to build and enhance automation tasks. You can drag and drop these commands into the Task Actions List panel, move commands, and edit commands in bulk, if necessary.

Using the Task Actions List

For your convenience, the Task Editor can display multiple tasks at once using tabs at the top of the Task Actions List. Using these tabs, you can open multiple tasks and edit them side-by-side, making copy/paste and reuse very easy. This also makes building larger tasks easier: you can record smaller processes, refine them, and then combine them into one large, end-to-end complete process. For more information, see Using the Task Actions List.

Using Error View and Variable Manager

You can use the Error View tab to manage errors that might occur in your task. The view is automatically updated as you make changes to the task. The Error View panel is useful when you want to track the progress of longer tasks and debug them when errors occur. Use the Error View panel in conjunction with the Error Handling capability to properly debug your tasks. For more information, see Debugging Tasks.

The Variable Manager panel is useful when using several variables in your tasks. The Variable Manager displays the local variables that are defined by the user for a particular task. You can use the Variable Manager to add, modify, and copy variables within automation tasks. For more information on how to use the Variable Manager, see Using the Variable Manager.

Using the Actions Buttons

At the top of the Task Editor window, several action buttons are available.

The 'Normal/Visualize' buttons enable you to switch views between task details (Normal) and a screen shot view of your task (Visualize).

On the same bar, actions are available for manipulating commands in the Task Actions List. For more information on these buttons, see Using the Task Actions List.

Above the Task Actions List, action buttons are provided for enabling debugging mode, and setting SnapPoints.

Using Filters

You can use the Filters bar to manage long tasks. When selecting these check boxes, the commands are made visible in the Task Actions List. Using these filters can make isolating problems and viewing specific commands easier.

For more information on using the Task Editor Filters, see Using Filters in the Task Editor.

Editing a Task Using the Keyboard

If you prefer not to use the mouse to drag and drop commands, Automation Anywhere makes it possible to insert commands in your task by using keyboard strokes. The Commands panel supports using Arrows, Tab, and Enter keys.

To open command windows, press the Enter key and complete the steps to specify command parameters.

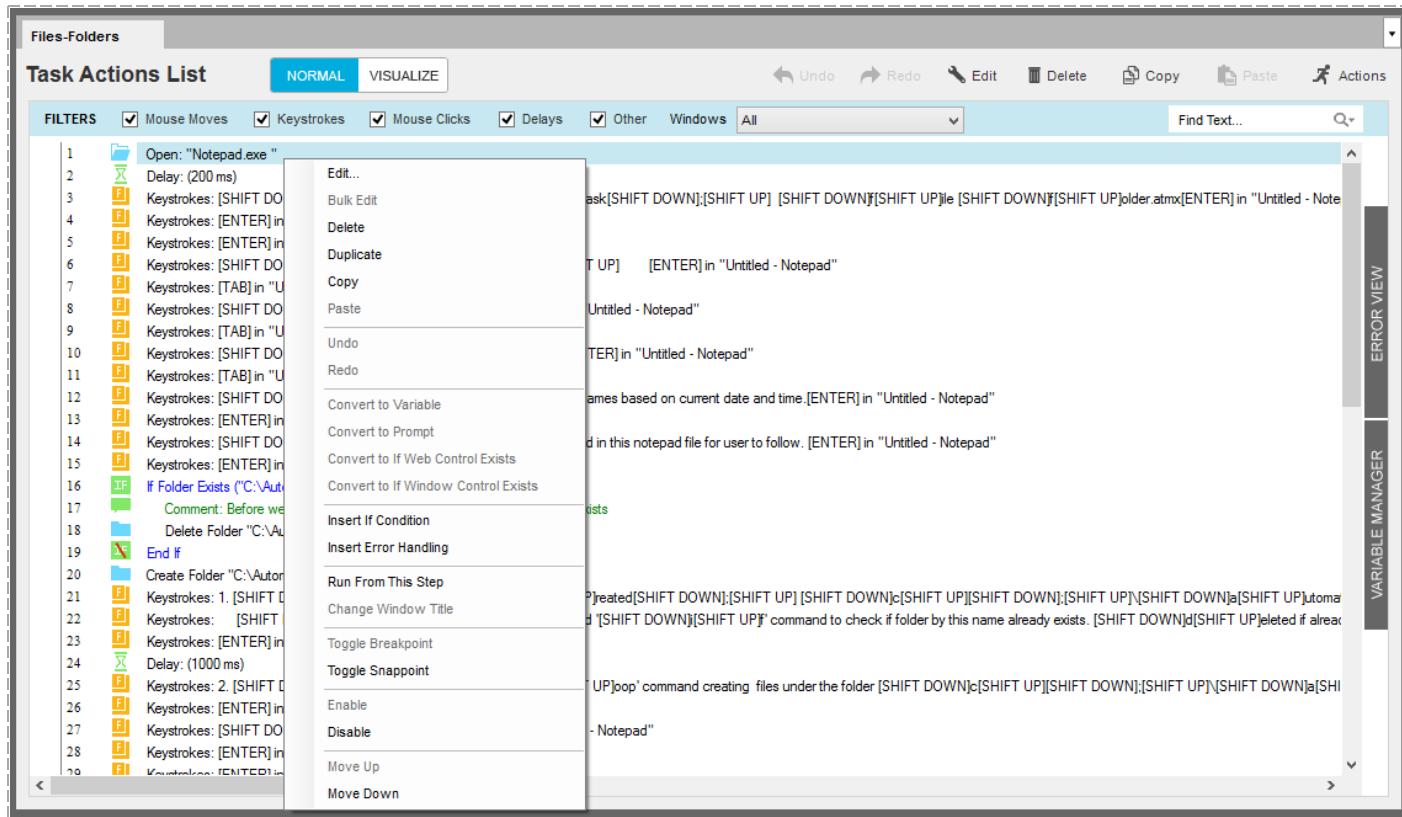This capability is useful for people who prefer to use quick keyboard shortcuts.

> Tip: Type the first letter of a command to highlight it and ease navigation to that command. For example, typing an "e" will highlight the Email Automation command.

Also See...

- Using the Task Actions List
- Using Filters in the Task Editor
- System Variables

Using the Task Actions List

The Task Actions List is the main workspace for building your automation tasks.

You can insert commands by dragging them from the Commands list. After you insert commands, you can modify them using the following options:

1. To edit the parameters (settings) for a command, double-click on the command.

2. To take action on the command, such as copying it, right-click on the command to view the options.

3. The Task Actions List bar provides buttons for manipulating commands. The following options are available:

   - **Edit:** Opens the command settings window to enable you to edit them.
   - **Delete:** Removes the command from the Task Actions List.
   - **Copy:** Copies a command for use in another location of the task.
   - **Paste:** Pastes a copied command at the location below the cursor.
   - **Actions:** Click on the Actions button to select one of the following actions:

     a. **Bulk Edit:** Make changes to multiple **Delay** or **Keystroke** commands simultaneously.

     b. **Duplicate:** Duplicate a command in the location below the selected command.

     c. **Convert to Variable:** Convert a **Keystroke** command to a variable.

     d. **Convert to Prompt:** Convert a **Keystroke** command to a prompt message.

     e. **Convert to if Web Control Exists:** Runs the command if the specified Web control exists.

     f. **Convert If Windows Control Exists:** Runs the command if the specified Windows control exists.

     g. **Insert If Condition:** Inserts an **IF** condition for a selected block of commands.

     h. **Insert Error Handling:** Inserts an **Error Handling** command for a selected block of commands.

     i. **Change Window Title:** Change the name of a window title to one or more application windows.

     j. **Run from this Step:** Runs the task when encountering this command.

     k. **Enable:** Activates a command.

     l. **Disable:** Disables a command.

m. **Move Up:** Moves a command up in the Task Action List.

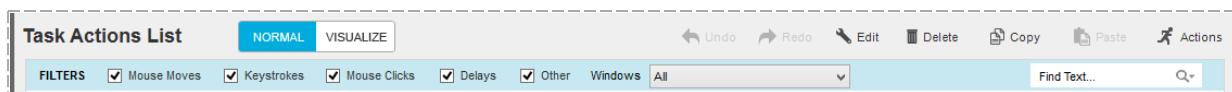n. **Move Down:** Moves a command down in the Task Action List.

Also See...

- Using the Task Editor

- Using Filters in the Task Editor

Using Filters in the Task Editor

By default, automated tasks are shown in the Automation Anywhere Task Editor with all commands and actions in chronological order (sorted by time). However, to make working with tasks easier, it provides useful filters to enable you to customize your view. Filters are particularly helpful when working with larger tasks.

Filters do not modify a task; rather, they enable you to focus on specific commands within a task without needing to modify the entire task.

To view or hide particular commands in a task, select or deselect the filters in the Filter Bar in the Task Actions List.

The commands that you can filter include:

- **Mouse Moves:** Display or hide all mouse move actions from the editor.
- **Keystrokes:** Display or hide all Insert Keystrokes commands and actions from the editor.
- **Mouse Clicks:** Display or hide all mouse click actions from the editor.
- **Delays:** Display or hide all Delay commands from the editor.
- **Other:** Display or hide all commands from the editor that are not mouse moves, keystrokes, mouse clicks or delays.
- **Windows:** View all actions by application window. The drop-down list displays all application windows that are used in the task.

Using the Find Text Search Field

Use the Find Text search field to search within a task for names, text, variables, and other items. This can be helpful when editing longer tasks.

Using the Windows Filter

You can use the Windows filter when a task involves two or more applications, such as the Calculator, Notepad, and Explorer. Use the Windows filter when you want to view actions sorted by application.

In the drop-down list, select ALL, NONE, or the application title for which you want to view commands.

Also See...

- Using the Task Actions List

Active Directory Command

Available in Enterprise edition only.

Use this command to manage your Active Directory and create, modify, and delete users in the system with centralized control.

Overview

An Active Directory is a directory service that is provided by Microsoft. This facility assists the Administrator in managing users across a group or wider organization.

> 💡Tip: All fields for the Active Directory command support the use of variables.

Automation Anywhere Enterprise uses LDAP (Lightweight Directory Access Protocol) to read from and edit users in the Active Directory. The Server name and Domain name are combined to create an LDAP path, which is used to connect to the Active Directory.

Establishing a Connection with Active Directory

To manage users in the Active Directory, ensure that a connection is established between the Administrator and the Active Directory server. Users who are logged into the Active Directory should have privileges for connecting to the Active Directory server and viewing the Server name, Domain name, user name and password details.

Sub-Commands

The Active Directory command provides the following sub-commands:

- Create User
- Modify User
- Create Group
- Modify Group
- Create Object
- Modify Object
- Search
- Get Property

Using the Sub-Commands

Follow these steps to insert Active Directory commands.

- **Create User Command:**

  This command creates a new user in the location specified in the Active Directory connection string. Follow these steps to insert the Active Directory - Create User command.

  1. Double-click or drag the Active Directory - Create User command to the Task Actions List pane. The Active Directory main window is displayed.

2. To establish a connection with the Active Directory, specify the Parent path in the Enter LDAP Path field.

3. If you do not know the Parent path, use the browse button to locate it.

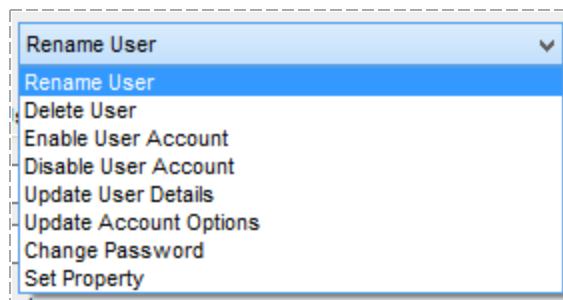4. The Active Directory Object Browser window is displayed.

5. Enter the details for a new user. Check the password option check boxes as needed.

6. Click Save to complete the process.

- **Modify User Command:**

    This command enables you to modify or rename users in the Active Directory. Follow these steps to insert the Active Directory - Modify User command.

    1. Double-click or drag the Active Directory - Modify User command to the Task Actions List pane. The Active Directory main window is displayed.

    2. Select one of the options in the 'Select User Action' drop-down menu.

    A. **Rename User:** Use the check boxes in the 'Enter Modify User Details to rename the user logon name and/or the user account name.

    B. **Enable User Account:** Enables a user account.

C. **Disable User Account:** Disables a user account. Select the User Name or Logon Name to disable either one, in the 'Enter Modify User Details' section.

Select User Action:

Select Action: Disable User Account

Enter Modify User Details:

⦿ User Name   ○ Logon Name   John Smith
e.g. John Smith

D. **Delete User:** Deletes the user from the Active Directory. Select the User Name or Logon Name to delete either one, in the 'Enter Modify User Details' section.

Select User Action:

Select Action: Delete User

Enter Modify User Details:

○ User Name   ⦿ Logon Name   john.smith123
e.g. john.smith123

E. **Update User Details:** Use this option to update the user's details.

F. **Update Account Options:** Use this option to set account attributes for the user. Select from four attributes: User must change password at next login, Is Active, User cannot change password, and Password never expires.

G. **Change Password:** Use this option to change a user's password.

H. **Set Property:** Use this option to assign a value to a user property , view the current details and update them.

💡Tip: Alternately, if you are unsure of the 'User Name' that has to be modified, click the browse button in 'Enter LDAP Path' section. You will be guided to the 'Active Directory Object Browser'; select a user from the list to perform any of the above 'User Actions'.

- **Create Group Command:**

  This command enables you to create a group in the Active Directory. Follow these steps to insert the Active Directory - Create Group command.

  1. Double-click or drag the **Active Directory - Create Group** command to the Task Actions List pane. The Active Directory main window is displayed.
  2. Select options for GroupName, Description, Group Scope and Group Type.

Modify Group Command:

This command enables you to modify a group in the Active Directory. Follow these steps to insert the Active Directory - Modify Group command.

1. Double-click or drag the **Active Directory - Modify Group** command to the Task Actions List pane. The Active Directory main window is displayed.

2. Select options for: Rename Group, Delete Group, Add Users To Group, Remove Users From Group and Set Property.

A. **Rename Group:** Use this option to rename an existing group.

B. **Delete Group:** Use this option to delete an existing group.

C. **Add Users to Group:** Use this option to add additional users to a group. A user list is displayed. Use the Add, Remove and Edit buttons and click on Add Users to the group.

Select Group Action:

Select Action: Add Users To Group ⌄

Enter Modify Group Details:

Group Name:

User List:

AA001	LDAP://192.168.2.60/CN=AA001,DC=Test,DC=COM
AA002	LDAP://192.168.2.60/CN=AA002,DC=Test,DC=COM
AA003	LDAP://192.168.2.60/CN=AA003,DC=Test,DC=COM

Add

Remove

Edit

💡 TIP : Press F2 to insert variable.

a. **Adding/Editing/Removing multiple users in a Group:** You can select a user or enter multiple users to add/edit/remove in/from a group from within the group list of the 'Active Directory Object Browser' Window.

Input the user names in the 'Enter user(s)' text box. If assigned, you can also insert variables.

Note: Multiple user names/variables must be separated with a ';' (semi-colon).

Active Directory Object Browser   ?

Enter Server Name: 192.168.2.60

Enter Domain Name: TASERVER.COM

Login: John.Smith   Password: *********   Go

DC=Test,DC=COM
  Computers
  Domain Controllers
  Engineering
  Finance
  ForeignSecurityPrinc
  Managed Service Ac
  Mktg
  Program Data
  PWD
  Sales
  System
  TestOU
  Users

Name	Description	Type

Enter User(s):

John Gray; New_AD_User

e.g. user1; user2; user3

☐ The above names are Logon Names.

💡 TIP : Press F2 to insert variable.

Save   Cancel

b. **Adding Logon Names as User(s):** If your Active Directory comprises users with same names, you can add 'Logon' names instead as 'User(s)'. To do so, select the option 'The above names are Logon Names' in the 'Active Directory Object Browser' window. Optionally, insert variable(s), if assigned.

c. **Editing Logon/User Names:** You can edit your Logon Name, User Name and even the LDAP string, if input incorrectly.Select the User, Click 'Edit' and you shall be guided to the 'Active Directory Object Browser' window, wherein you will be able to input the necessary changes. Click 'Save' once done.

> Note: When editing a User Name, you can also select the 'Logon Name' option.

D. **Remove Users from Group:** Use this option to remove users from a group. A user list is displayed. Use the Add and Remove buttons and click on Remove Users.

E. **Set Property:** Use this option to assign a value to a group property.

- **Create Object Command:**

    This command enables you to create a new object, either for Computer or for Organizational Unit, in the Active Directory. Follow these steps to insert the Active Directory - Create Object command.

    1. Double-click or drag the **Active Directory - Create Object** command to the Task Actions List pane. The Active Directory main window is displayed.
    2. Select the option for creating either a new Organizational Unit or a new Computer by using the radio button.

- **Modify Object Command:**

    This command enables you to modify an object in the Active Directory. Follow these steps to insert the Active Directory - Modify Object command.

    1. Double-click or drag the **Active Directory - Modify Object** command to the Task Actions List pane. The Active Directory main window is displayed.
    2. Select options for: Rename Object, Delete Object, Move Object, and Set Property.
        A. **Rename Object:** Use this option to rename an existing computer or organizational unit.
        B. **Delete Object:** Use this option to delete an existing computer or organizational unit.
        C. **Move Object:** Use this option to change the location of an existing computer or organizational unit.
        D. **Set Property:** Use this option to assign a value to an object property.

Active Directory

Select Command:

- ○ Create User
- ○ Create Group
- ○ Create Object
- ○ Search
- ○ Modify User
- ○ Modify Group
- ⦿ Modify Object
- ○ Get Property

Enter LDAP Path:

Parent Path: `LDAP://192.168.2.61/DC=TASERVER,DC=COM`   [...]

e.g. LDAP://DomainServer/CN=Users;DC=Domain,DC=Com

Login User: `Administrator`   Login Password: `************`

Select Object Action:

Select Action: `Rename Object` ▾

- Rename Object
- Delete Object
- Move Object
- Set Property

Enter Modify O

Object Name: [   ]

New Name: [   ]

Object Type: ⦿ Computer   ○ Organizational Unit

💡 **TIP :** Press F2 to insert variable.

[Save] [Cancel]

Search Command:

This command enables you to search the Active Directory for users of a group or run a query on the Active Directory. Follow these steps to insert the Active Directory - Search command.

1. Double-click or drag the **Active Directory - Search** command to the Task Actions List pane. The Active Directory main window is displayed.

2. Select one of the following options from the drop-down window:

A. **Get All Users of a group:** Use this option to find all users that are members of a group, and stores them in a list variable. Depending on the return type of the list variable, it contains the user names or the LDAP paths for the members.

B. **Run Query:** Use this option to specify an Active Directory query and run it. The query results are stored in a list variable, either for object names or LDAP paths, depending on the option selected.

Active Directory ?

Select Command:

○ Create User   ○ Create Group   ○ Create Object   ◉ Search

○ Modify User   ○ Modify Group   ○ Modify Object   ○ Get Property

Enter LDAP Path:

Parent Path: `LDAP://192.168.2.61/DC=TASERVER,DC=COM`   `...`

e.g. LDAP://DomainServer/CN=Users;DC=Domain,DC=Com

Login User: `Administrator`   Login Password: `************`

Select Search Action:

Select Action: `Get All Users Of a Group` ⌄
`Get All Users Of a Group`
Enter Object De | `Run Query`

Object Name: `QA_Group`

Return Type: ◉ Object Name   ○ LDAP Path

Assign Value To Variable: `my-list-variable` ⌄

💡 **TIP :** Press F2 to insert variable.

Save   Cancel

💡 Tip: For Active Directory (LDAP) queries, you may refer this link.

- **Get Property Command:**

  This command enables you to retrieve a specific property value for an object (user name, group name, computer, or organizational unit) to a variable. Follow these steps to insert the Active Directory - Get Property command.

  1. Double-click or drag the Active Directory - Get Property command to the Task Actions List pane. The Active Directory main window is displayed.

  2. Select the object type: user, group, computer, or organizational unit.

Active Directory

Select Command:

- ○ Create User
- ○ Create Group
- ○ Create Object
- ○ Search
- ○ Modify User
- ○ Modify Group
- ○ Modify Object
- ● Get Property

Enter LDAP Path:

Parent Path: `LDAP://192.168.2.61/DC=TASERVER,DC=COM` `...`

e.g. LDAP://DomainServer/CN=Users;DC=Domain,DC=Com

Login User: `Administrator`   Login Password: `************`

Select Object Type

Select Action: `User` ▾

- User
- Group
- Computer
- Organizational Unit

Enter Modify O

Object Name: [   ]

Property: [   ]

Assign Value To Variable: `Select Variable` ▾

💡 **TIP :** Press F2 to insert variable.

[Save] [Cancel]

Also See...

- Active Directory Command - Examples
- Printer Settings Command
- Services Command
- SNMP Command

App Integration Command

Available in Enterprise and Small Business editions only.

Use this command to integrate applications with your automated task and capture text data.

Overview

Automation Anywhere makes it easy for you to integrate any application with your automated tasks using simple drag-and-drop operations. The App Integration command supports a large number of applications, including Java and Windows applications.

Sub-Commands

The App Integration command provides the following sub-commands:

- Browsers, such as Microsoft Internet Explorer and Mozilla Firefox
- DOS command prompts
- Java applications
- Java Applets
- Telnet Unix Shell
- Windows applications
- Other

Using the Command

To insert an App Integration command in your task, follow these steps:

1. Double-click or drag one of the **App Integration** commands to the Task Actions List pane. The App Integration window is displayed.

2. Select the application type from the drop-down window.

3. Select a capture option. You can capture text data from any application window by using the following options:

   - **Capture Window:** Specify an application window title. When running the task, Automation Anywhere will capture the window as an image.

   - **Capture Area:** Specify a specific area of an application window to capture.

   - **Capture Scrollable Text:** Specify an area with scrollable text to capture all of the text in that area.

4. Select a window from which you want to capture the text.

5. Optionally specify a range of text, and assign the captured text to a variable.

The App Integration command is flexible, enabling you to specify a range of text to capture. Use the **Before** and **After** keywords to specify the range.

Example: In the text string:

```
"Version 5.1 (Build 2600.xpsp_sp3_gdr.090206-1234 : Service Pack 3) Copyright"
```

- Specify "Version" in the **Before** keyword field, and "Copyright" in the **After** keyword field.

- You can also specify that you want to trim the captured text to remove leading and trailing spaces.

- Use this command to capture scrollable text that you cannot normally copy from documents like PDFs.

   Note: Non-standard Windows controls are supported.

App Integration

Step 1 : Select application type

Windows Application

Step 2 : Select option

○ Capture Window   ○ Capture Area   ● Capture Scrollable Text

Step 3 : Select window

Document1 - Word   [Refresh]

[Captured Scroll Area]   [View Captured Text]

Step 4 : Assign the captured text to a variable when the task runs

☑ Filter the captured text

Before (Optional)   After (Optional)

💡 **TIP :** Press F2 to insert variable.   💡 **TIP :** Press F2 to insert variable.

Example :
To extract $249 from 'Price: $249 xyz'
Specify "Before" = Price: & "After" = xyz

☑ Trim the captured text

Assign the value to an existing variable   Select Variable

Image Preview :

Captured Text :

[Save]   [Cancel]

Also See...

- App Integration Command - Examples
- Database Command
- Email Automation Command
- Excel Command
- OCR Command
- PDF Integration Command
- Read from CSV/ Text Files Command
- Terminal Emulator Command
- XML Command

Clipboard Command

Available in Enterprise and Small Business editions only.

Use this command to automate Windows clipboard activities.

Overview

The Windows Clipboard is a very common application, and is used for a wide variety of purposes. When you copy text to the Clipboard (for example, by using **Ctrl+C**), you can then paste this text in another application (by using **Ctrl+V**). Automation Anywhere provides a system variable, named *$Clipboard$*, that you can use to retrieve text that has been copied to the Clipboard.

Sub-Commands

The Clipboard command provides the following sub-commands:

- Clear Clipboard
- Assign to Clipboard
- Assign from Clipboard
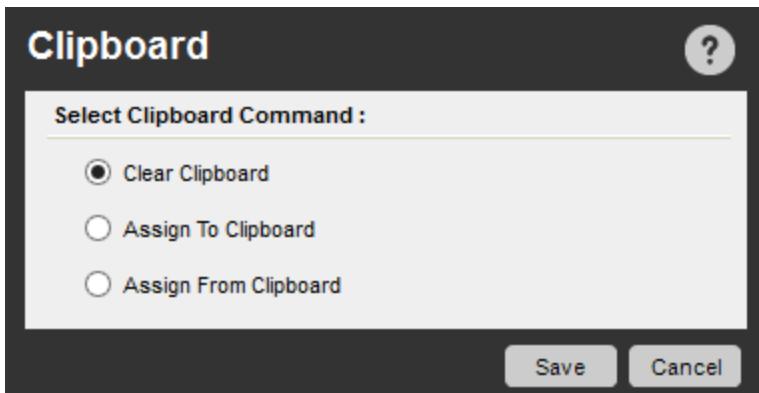
Using the Sub-Commands

To insert a Clipboard command in your task, follow these steps:

1. Double-click or drag one of the **Clipboard** commands to the Task Actions List pane. The Clipboard window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.

Clear Clipboard Command:

Use this command to clear the contents of the Clipboard.

Assign To Clipboard Command:

Use this command to assign the value of any variable that has been created using the Variable Manager to the Clipboard. You can then access this value by using the *$Clipboard$* system variable.

Assign From Clipboard Command:

Use this command to assign the value contained in the Clipboard to any value type variable that has been created using the Variable Manager.

Also See...

- Clipboard Command - Examples
- Using the Variable Manager
- Using Variables
- Value Type Variables
- System Variables
- Comment Command
- Error Handling Command
- Image Recognition Command

- Screen Capture Command
- Play Sound Command
- String Operation Command
- Variable Operation Command

Comment Command

Use this command to insert comments in your automation tasks to provide additional information about the task. Comments are ignored when the task runs.
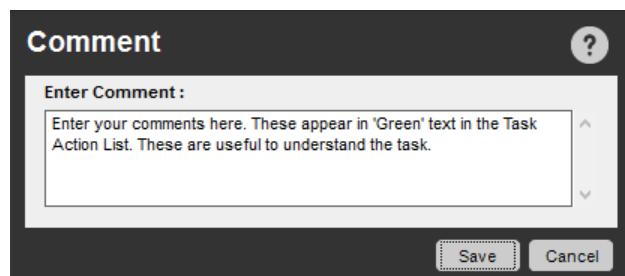
Comments are useful in annotating the task steps. Some people use comments to extensively document details about their tasks. Some people use fewer comments just as reminders.

Inserting a Comment

To insert a comment, follow these steps:

1. Open the task in the Task Editor.

2. In the commands list, drag a Comment command to the Task Actions List, in the location where you want the comment to be displayed. Or, you can double-click the Comment command to insert it at the current location.

3. Type your comments in the Comment text box provided.

4. Click Save.

5. A comment is inserted in the task. You can copy this comment to add multiple comments, or move the comment to another position in the task simply by dragging it.

The comment is displayed in green in the Task Actions List, and is always saved as a single line. Multiple-line comments are displayed as a single line when the comment is saved.

Note: Though a command, a comment line is not executed.

Sample Comments

Also See...

- Comment Command - Examples
- Clipboard Command
- Error Handling Command
- Image Recognition Command
- Screen Capture Command
- Play Sound Command
- String Operation Command
- Variable Operation Command

Database Command

Available in Enterprise and Small Business editions only.

Use this command to integrate databases with your automated task and move or capture data.

Sub-Commands

The App Integration command provides the following sub-commands:

- Connect
- Disconnect
- SQL Query
- Insert/Update/Delete
- Run Stored Procedure

Overview

Automation Anywhere enables you to transfer data between a database and other applications. You can import or export data, in various formats, and convert data from one format to another. For example, you might want to transfer and convert data while completing a data migration project.

You can also transfer data from a custom database to a Microsoft Excel spreadsheet, or from one database to another in pre-defined formats.

Restriction: For the Database command, Microsoft Access 2007 and Access 2010 users are required to use Microsoft Office 12.0 Access Database Engine OLE DB Provider.

Using the Sub-Commands

To insert a Database command in your task, follow these steps:

1. Double-click or drag one of the **Database** commands to the Task Actions List pane. The Database window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.

Connect Command:

Use this command to connect to a database by providing a session name and a connection string.

To complete the connection string, click the browse button to display the Data Link Properties window. Select the database provider and type the full path of the data source. When done, click on Save.

> Tip: When using the 'Connect' command to select a target database, click on the 'Test Connection' button in the 'Data Link Properties' window to ensure that the database connection is working.

Disconnect Command:

Use this command to disconnect from a database.

SQL Query Command:

Use this command to issue SQL queries using a Select statement.

Tip: If you must use wildcards when executing SQL queries, insert "*" if using MSAccess or "%" in MSSQL.

Optionally, you can:

- Specify the number of records to fetch
- Specify a timeout value for the query
- Export the data file as a CSV (comma-separated) file in a specified location. Exporting data to CSV files support ANSI, UNICODE, and UTF8 encoding.

Insert/Update/Delete Command:

Use this command to specify an Insert, Update, or Delete operation by using a Select statement.

Run Stored Procedure Command:

Use this command to run an existing procedure that is stored in your database. You can add and update by providing Input and Output types of parameters.

You can export the data file to a CSV file in a specified location.

Sample Scenario: Transferring Data from a Database

The following code snippet illustrates how a task can use a LOOP command to transfer data from a database to a target application.

Database_SQ...

Task Actions List   NORMAL   VISUALIZE   ↩ Undo   ↪ Redo   🔧 Edit   🗑 Delete   📋 Copy   📋 Paste   🏃

FILTERS   ☑ Mouse Moves   ☑ Keystrokes   ☑ Mouse Clicks   ☑ Delays   ☑ Other   Windows   All ▼   Find Text...

14	Comment: Database CONNECTION..........
15	Connect to "Provider=Microsoft.Jet.OLEDB.4.0;Data Source=$RequiredFilesFolder$\DB_Student.mdb;Persist Security Info=False" Session:'Access Database'
16	Log to File: Access Database Connection ....PASS in "$RequiredFilesFolder$\$LogFilePath$"
17	Connect to "Provider=SQLOLEDB.1;Integrated Security=SSPI;Persist Security Info=False;Initial Catalog=Books;Data Source=193.193.193.39\SQLEXPRESS" Session:'SQLServer Database'
18	Log to File: SQL Server Connection ....PASS in "$RequiredFilesFolder$\$LogFilePath$"
19	Comment: * * * * * * * * SELECT Query..........
20	Comment: ====================== ACCESS DB ============
21	Execute SQL Statement: 'Select * from student' Session:'Access Database'
22	Log to File: ------------------- Current Status of Access database ------------------- in "$AccessDataPath$"
23	Start Loop "Each row in SQL Data Set Session: Access Database"
24	Log to File: $Dataset Column(1)$ - $Dataset Column(2)$ - $Dataset Column(3)$ in "$AccessDataPath$"
25	End Loop
26	Log to File: Access - SELECT Query ....PASS in "$RequiredFilesFolder$\$LogFilePath$"
27	Comment: ======================= SQL ============
28	Execute SQL Statement: 'select * from books' Session:'SQLServer Database'
29	Log to File: -------------------- Current Status of SQL Server database -------------------- in "$SQLServerDataPath$"
30	Start Loop "Each row in SQL Data Set Session: SQLServer Database"
31	Log to File: $Dataset Column(1)$ - $Dataset Column(2)$ - $Dataset Column(3)$ in "$SQLServerDataPath$"
32	End Loop
33	Log to File: SQL Server - SELECT Query ....PASS in "$RequiredFilesFolder$\$LogFilePath$"
34	Comment: ======================= INSERT Query......... ============
35	Comment: ======================= ACCESS DB ============
36	Execute SQL Statement: 'Insert into student values('S006' , 'Nick Smith', 'Adventure')' Session: 'Access Database'
37	Execute SQL Statement: 'Select * from student' Session:'Access Database'
38	Log to File: -------------------- Status of Access Database after INSERT Query -------------------- in "$AccessDataPath$"
39	Start Loop "Each row in SQL Data Set Session: Access Database"
40	Log to File: $Dataset Column(1)$ - $Dataset Column(2)$ - $Dataset Column(3)$ in "$AccessDataPath$"
41	End Loop

💡 Tip: Use LOOP commands in conjunction with the Database commands to transfer data from a database to another application. For example, you might loop 'Each row in an SQL Query dataset'.

Also See...

Delay/Wait Commands

Use the delay command to add a delay at a particular point in an automated task. Or the wait command to insert a wait condition in an automated task.
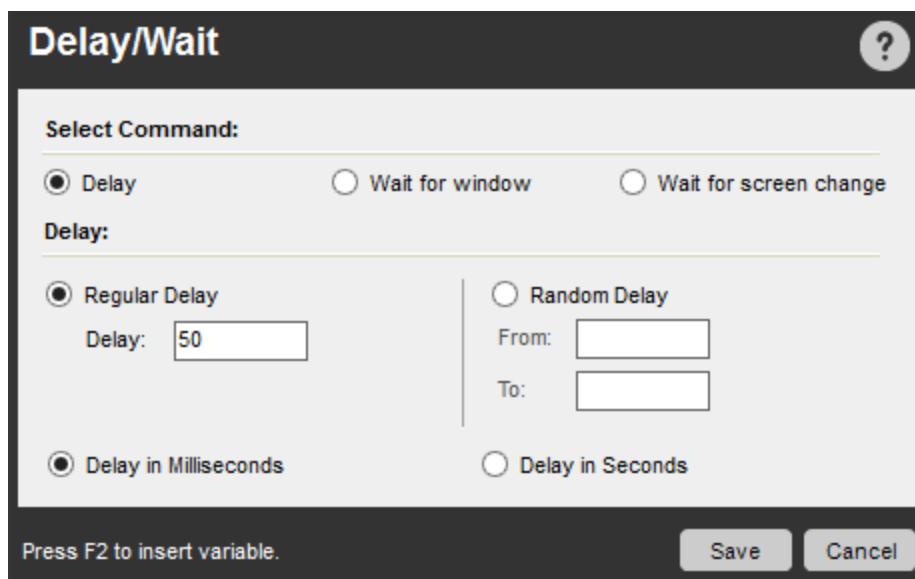
Delay Command

- In certain cases, you might want a task to wait a certain amount of time before moving to the next command. You can use a Delay command to make the task wait, and you can specify the amount of wait time.
- The Loop command is one of the more powerful commands in Automation Anywhere. When using loops, inserting delays can be helpful for performing the subsequent actions in the task, optimizing the timing that occurs.

Using the Delay Command

To insert a Delay command, follow these steps:

1. Open the task in the Task Editor.

2. In the commands list, drag a Delay command to the Task Actions List, in the location where you want to the task to delay before moving to the next command. Or, you can double-click the Delay command to insert it at the current location.

3. A Delay command is inserted in the task. You can copy this command to add multiple delays, or move the delay to another position in the task simply by dragging it.

4. Optionally specify that the delay be in milliseconds or seconds. Or, insert a random delay by providing a range of numbers that Automation Anywhere can use to generate a random number for the delay time.

> Note: You can assign a variable to 'Regular Delay > Delay' and 'Random Delay > From and To' options.

Wait Command

- For some applications, automating actions might require waiting for a screen to change, or another window to open. Let's say you want to wait for the contents of a screen (or an area within your application) to change before performing the next set of actions (such as commands, mouse clicks, or keystrokes). This is useful in many application windows where the text is not a selectable format.
- In this case, you might mark the area by using the capture button. An image will be displayed in the image preview so that you can verify the captured portion. Keep in mind that the sample image is not used for comparison while the task is running; it is only used to verify that the image was captured correctly. When the task runs, a screen shot of the area is captured, using the coordinates relative to the screen or window that you've specified. Here, let's call this image 'imageA.'
- Automation Anywhere now waits for the amount of time to pass that you've specified, then begins comparing the images on the page to imageA. It continues to compare the image in the marked rectangular section to imageA, and when differences occur, the comparison stops and the task proceeds to the next command.
- However, if the images never differ within the specified time, it either stops the task or proceeds to the next command.
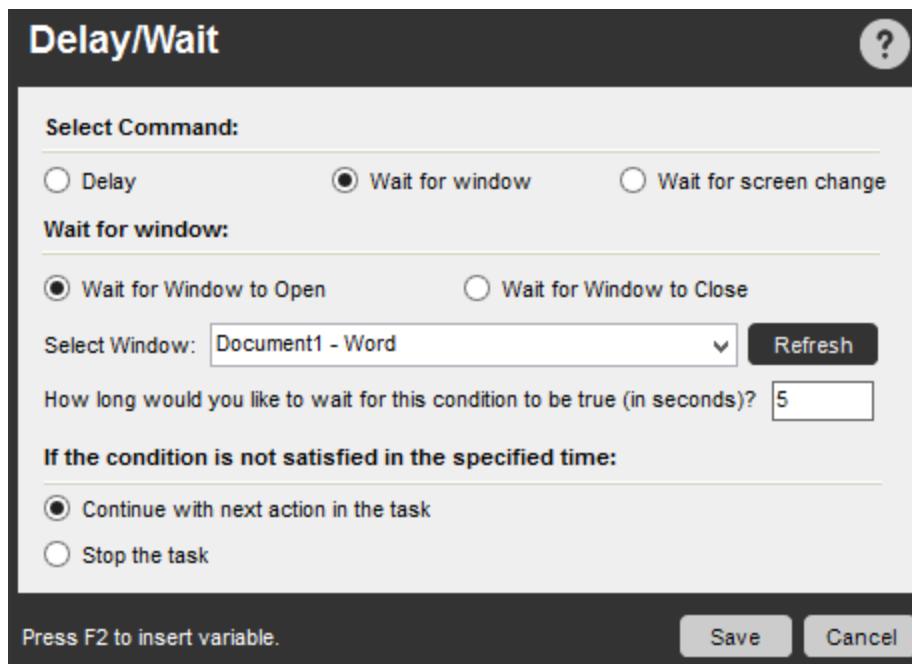
Sub-Commands

The Wait command provides the following sub-commands:

- Wait for Window
- Wait for Screen Change

Wait For Window Command
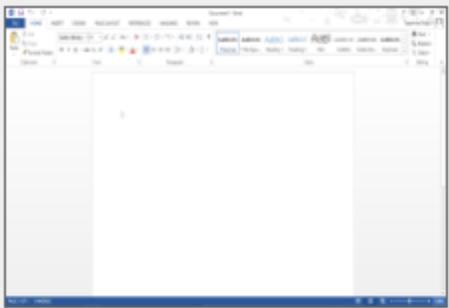
Use the Wait For Window command to wait for a specified window to open or close before the task performs the next command.

Wait For Screen Change Command

Use this command to wait until a rectangular image on the screen changes before the task performs the next command.

> Note: You can assign a variable when selecting a window in 'Wait for Window' and 'Wait for
> Screen Change'.

Also See...

- Delay Command - Examples
- Wait Command - Examples
- Loop Command
- Pause Command
- Run Task Command
- Stop Task Command

Email Automation Command

Available in Enterprise and Small Business editions only.

Use this command to automatically manage your email messages with your Automation Anywhere task.

Overview

You can use the Email Automation command to intelligently automate your mail server functions and handle incoming email messages. Specifically, you can automate the following activities:

- Manage your incoming email messages and download attachments to specific folders.
- Clear unwanted email messages.
- Communicate with your mail server and monitor email activity.
- Extract email data (such as Subject, Message, From:, To: , CC:, and attachments) to applications (Excel, Word, Notepad, and others) or folders on your computer.
- Select either IMAP or POP3 servers.
- Organize email messages while downloading them, based on status (All, Read, and Unread).
- Select either HTML and Plain text formats for your email messages.

Sub-Commands

The Email Automation command provides the following sub-commands:

- Get All Messages
- Delete All Messages
- Delete Message

Using the Sub-Commands

To insert an Email Automation command in your task, follow these steps:

1. Double-click or drag one of the **Email Automation** commands to the Task Actions List pane. The Email Automation window is displayed.
2. Specify the email server information in the **Email Server Details** fields. Provide a Host name, user name, and password. Optionally check the **Server uses secure connection(SSL)** check box for secure connections.
3. Select a server type. Depending on the email server in your environment, select either IMAP or POP3.
4. Select an email status (All, Read, or Unread).
5. Select a message format. You can download messages in HTML or plain text format.
6. Optionally specify a location to which you want to download the email data.
7. Click Save to save the command to your automation task.

Get All Messages Command

Use this command to download all incoming email messages from the mail server to a specified folder on your computer.

The following screen shot shows how you can download messages from a POP3 server in HTML format.

Email Automation

Select Option :

- ● Get All Messages (Leaves messages on server)
- ○ Delete All Messages   ○ Delete Message

Email Server Details :

Host Name : [   ]

e.g. mail.MyIsp.com   💡 **TIP :** Press F2 to insert variable.

User Name : [   ]

e.g. user@MyIsp.com   💡 **TIP :** Press F2 to insert variable.

Password : [   ]

💡 **TIP :** Press F2 to insert variable.

☑ Server uses secure connection (SSL)

Server Type :

- ○ IMAP   ● POP3

Port Number : [143   ]   [Use Default]

💡 **TIP :** Press F2 to insert variable.

Message Format : ● HTML   ○ Plain Text

Save attachment in

[   ]   [...]

eg. C:\Myfolder\   💡 **TIP :** Press F2 to insert variable.

[Save] [Cancel]

Delete All Messages Command

Use this command to delete all email messages from the mail server.

Delete Message Command

Use this command to delete the most recent email message from the Inbox. Use a LOOP command with the Email Automation command.

Also See...

- Email Automation Command - Examples
- Database Command
- Excel Command
- Loop Command
- PDF Integration Command
- Read from CSV-Text Command

- Terminal Emulator Command
- XML Command

Error Handling Command

Use this command to handle errors while running a task, to aid in debugging the task.

Overview

By using the Error Handling commands, you can isolate errors that occur when running your tasks.

When an error occurs, you can continue the task after the error, or stop the task.

Sub-Commands

The Error Handling command provides the following sub-commands:

- Begin Error Handling
- End Error Handling

Error Handling Options

Automation Anywhere provides the following options for error handling when an error occurs:

1. **Take Snapshot:** In case of any error, take a snapshot of the screen.
2. **Run Task:** Run another task when the current task encounters an error.
3. **Log data into file:** Log the error into a file.
   - For example, you can log the Error Line No. and Description, then use the system variables *$Error Line Number$* and *$Error Description$*.
4. **Send Email :** Send an email when the error occurs. Optionally 'Attach Snapshot' and/or 'Attach Variable'. You can configure email settings from Tools > Options > Email Settings
   - On selecting 'Attach Snapshot', a snapshot of the screen is attached to the email.
   - On selecting 'Attach Variable', a text file that lists the values of all Variables (System Variables + User Defined Variables) is attached to the email.

     > Note: The System Variable *Excel Cell* will not be attached as the values input are dynamic.

5. **Variable Assignment:** Specify a value to be assigned to a variable and Set Task Status Pass or Fail depending on the Error handling action.
   - This is more useful when one wants to Continue running a task, and thus in that case the user wants to set a specific value to a particular variable.

Use any or all four of these options in combination. For example, you can run a task and log data to a file.

When any of the Begin Error Handling option is selected, End Error Handling is automatically inserted in the task editor. However, one can also specifically select End Error Handling action when the related actions are completed.

Also See...

- Error Handling Command - Examples
- Using the Variable Manager
- Using Variables
- Value Type Variables
- System Variables
- Variable Operation Command

Excel Command

Use this command to transfer data to and from Microsoft Excel spreadsheets.

Overview

The Excel command is one of the most powerful commands provided by Automation Anywhere. Microsoft Excel is one of the most highly used applications in the world today. Most operations that involve using spreadsheets require repetitive tasks that can be automated.

Many Excel automation scenarios have been implemented by companies worldwide. Some common scenarios include:

- Copying data from one Excel spreadsheet to another.
- Extracting data from an application to an Excel spreadsheet.
- Extracting data from a website to an Excel spreadsheet.
- Moving data from an Excel spreadsheet to another application.
- Transfer data from Microsoft Access to Excel.
- Transfer Excel data to a website form.
- Delete rows or columns in Excel.
- Compare two columns or two cells in an Excel spreadsheet.
- Sort data in Excel spreadsheets and delete duplicate rows.

You can also run Excel macros as part of complete, end-to-end processes.

> Note: The Excel command is only supported for Microsoft Office 2000 and later.

Sub-Commands

The Excel command provides the following sub-commands:

- Open Spreadsheet
- Close Spreadsheet
- Activate Sheet
- Get Cell
- Set Cells
- Go To Cell
- Run Excel Macro
- Delete Cells
- Save Spreadsheet
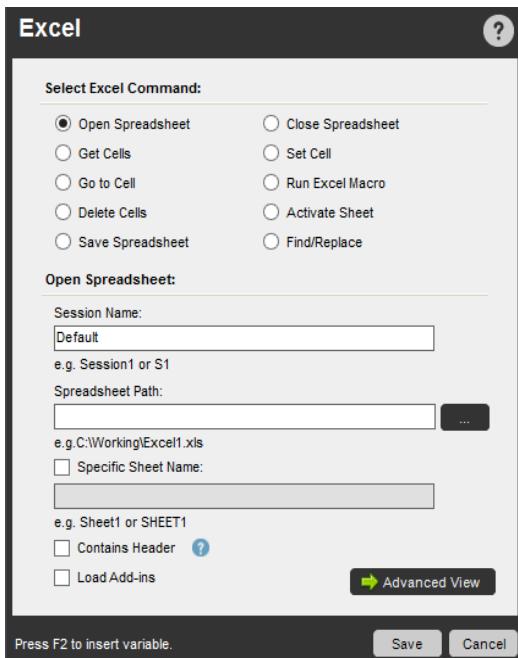- Find/Replace

Open Spreadsheet Command

Use this command to open an Excel spreadsheet. If your task involves working with many spreadsheets and workbooks, you can specify the session name, sheet name and even passwords for users with read only or edit rights. Also, fetch data from spreadsheets with headings.

Enabling Contains Header option - You can retrieve/fetch data from cells in spreadsheet(s) that contain headings in the **first row**. You can enable the option in Open Spreadsheet to be able to apply it to relevant sub-commands for that particular session.

Refer Excel Command - Contains Header for details.

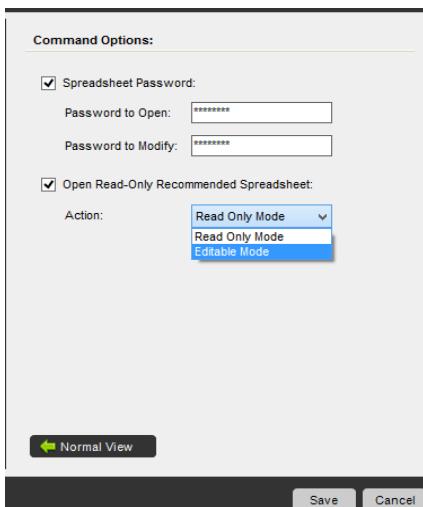- **Excel Cell System Variable -** has been introduce exclusively for the 'Contains Header' option; hence we recommended you use it when configuring Excel operations - Get Cells, Set Cell, Go to Cell, Find/Replace and Delete Cells.

> Note: For Automation tasks created in versions 8.1 and below, the Contains Header was applicable to Get Cells command only. Hence to ensure that those tasks run properly, it is recommended that you re-configure the command.

(Image as seen in Enterprise Edition 8.2)

Spreadsheet Password - Password options can be enabled in 'Advanced View'. You can open/edit your password protected spreadsheet(s)and workbook(s) by enabling the 'Spreadsheet Password' option. Specify the password(s) to open and/or modify your spreadsheet(s).

(Image as seen in Enterprise Edition 8.2)

If you wish, you can assign a variable to your password(s). The specified password(s) are secure as they are encrypted.
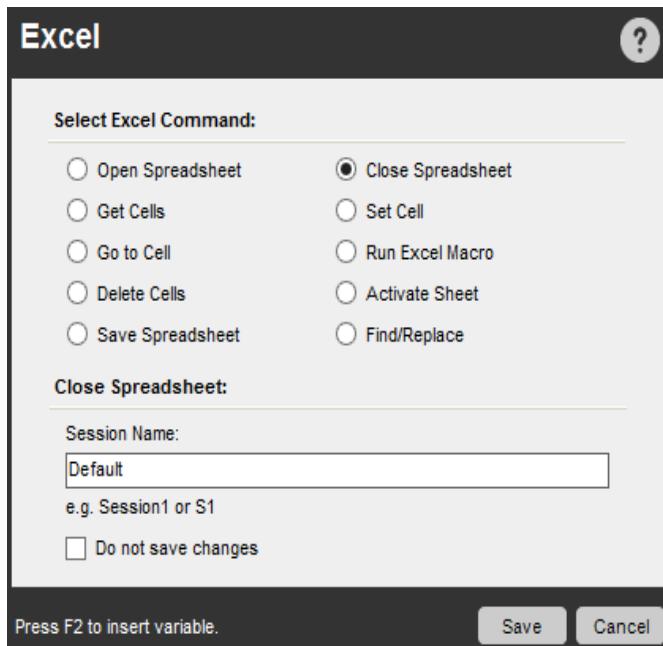
> 📝 Note: When enabling the 'Spreadsheet Password' option, ensure that you specify either the 'Read Only' or 'Editable' mode listed in the 'Open Read-Only Recommended Spreadsheet' option.

Open Read-Only Recommended Spreadsheet - You can also select 'Open Read-Only Recommended Spreadsheet' independently to open these in 'Read Only' or 'Editable' Mode.

- For instance, if your spreadsheet is *password protected* and *read only* you will be required to specify the 'Password to Open' *plus select* 'Read Only Mode'.
- If, however, you specify passwords to 'Open' and 'Modify', you have the option to open the spreadsheet in either 'Read Only' *or* 'Editable' Mode, as required.
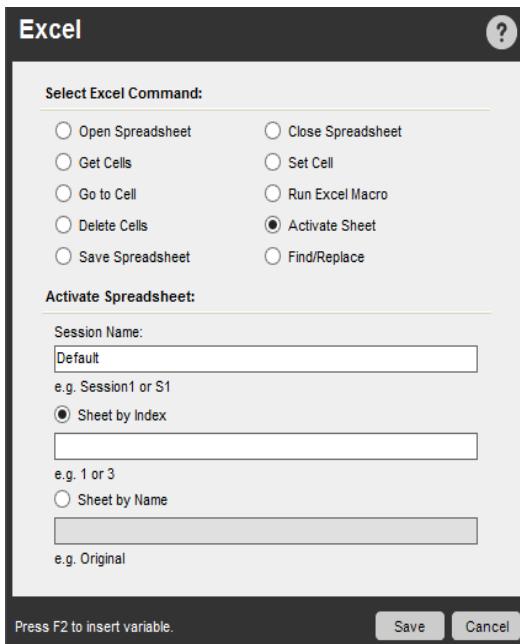
Close Spreadsheet Command

Use this command to complement the Open Spreadsheet command. It closes a spreadsheet that is previously opened using an Open Spreadsheet command. The command identifies the spreadsheet using the session name that was provided in the Open Spreadsheet command.

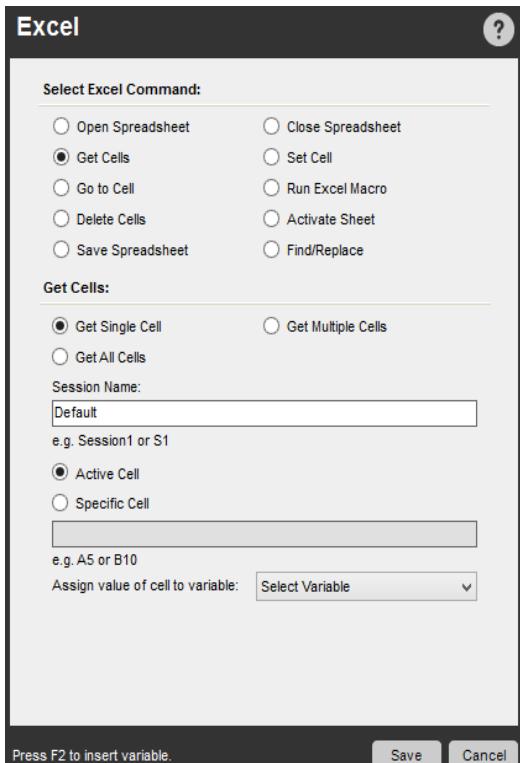*(Image as seen in Enterprise Edition 8.2)*

Activate Sheet Command

Use this command to activate a specific spreadsheet in an open workbook.

(Image as seen in Enterprise Edition 8.2)

Get Cells Command

Use this command to retrieve the values of the active cell or a specific cell, as well as a range of cells from the spreadsheet opened by the Open Spreadsheet command. When you retrieve values for an active or specific cell, you can assign the value to a variable.
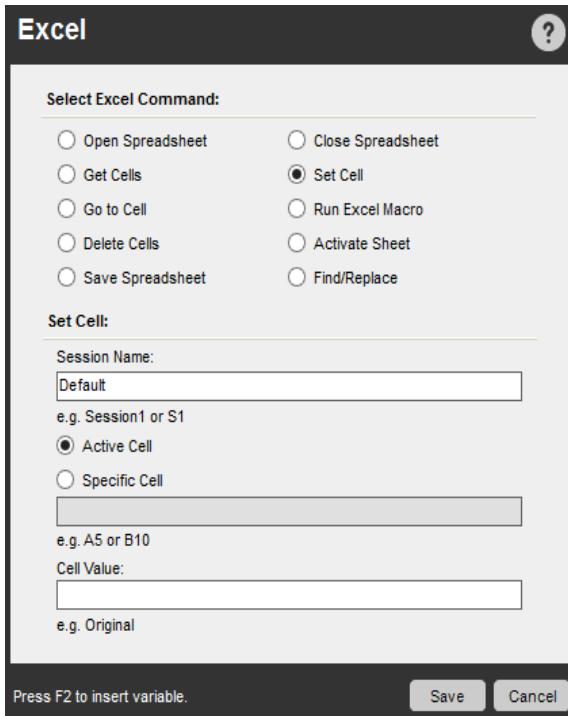

(Image as seen in Enterprise Edition 8.2)

When retrieving values from **multiple cells**, the values are assigned to an *$Excel Column (index)$* system variable. You can use this variable in subsequent commands, for example, to loop through each row in the spreadsheet.

Set Cells Command

Use this command to set the value of the active cell or a specific cell in an Excel spreadsheet previously opened using the Open Spreadsheet command. You can also use a variable to assign the value in the cell.
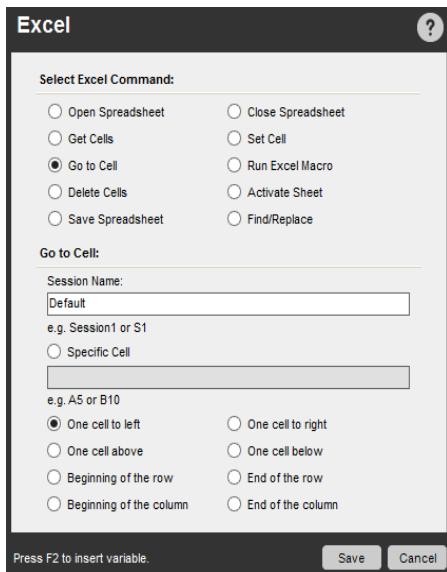

(Image as seen in Enterprise Edition 8.2)

Go to Cell Command

Use this command to move to a specific cell. The movement options include:

- Move one cell to the left or to the right of the active cell
- Move one cell above or below the active cell
- Move to the beginning or end of a row or column within a spreadsheet that was previously opened using the Open Spreadsheet command.
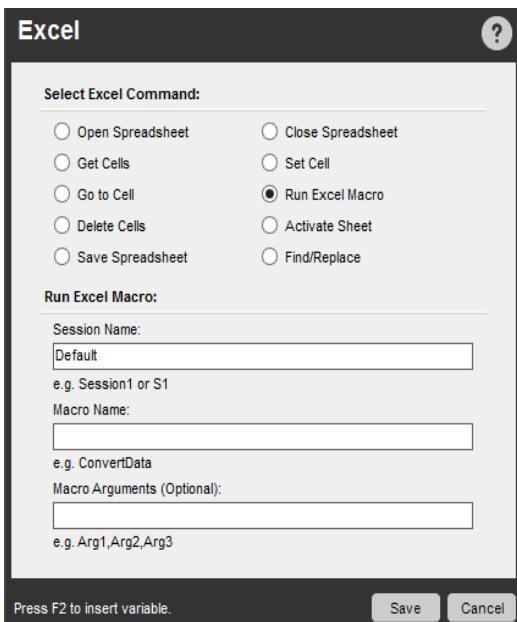
> Note: In **Active Cell, End of the row/column** refers to "end of the row/column where the data ends"; **not** the end of spreadsheet row/column.

(Image as seen in Enterprise Edition 8.2)

Run Excel Macro Command

Use this command to run an Excel macro that is stored within an Excel spreadsheet that was previously opened with the Open Spreadsheet command.
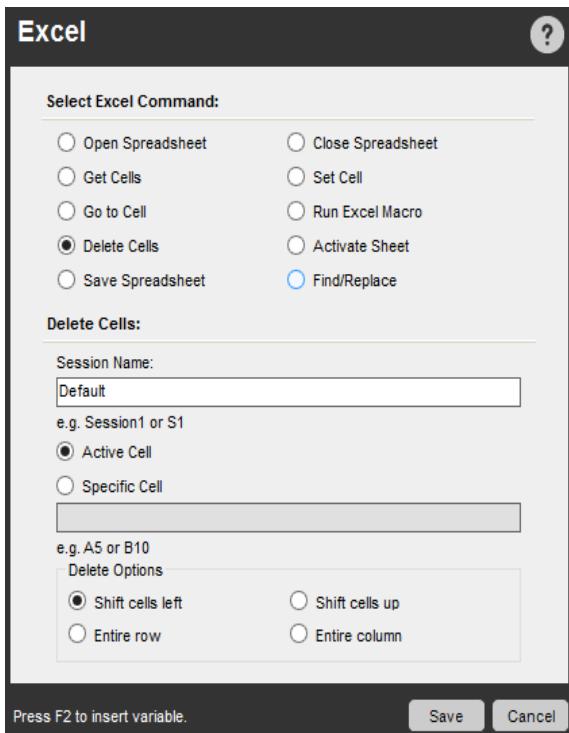

(Image as seen in Enterprise Edition 8.2)

Note: Automation Anywhere now enables you to Run Local and Global Macros seamlessly.
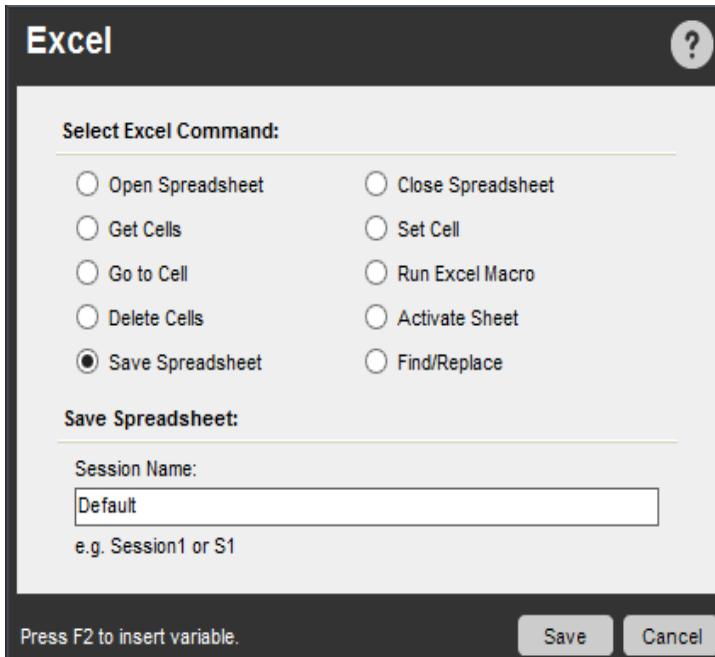
Delete Cells Command

Use this command to delete the values in the active cell or in a specific cell within an Excel spreadsheet that was previously opened using the Open Spreadsheet command.

(Image as seen in Enterprise Edition 8.2)
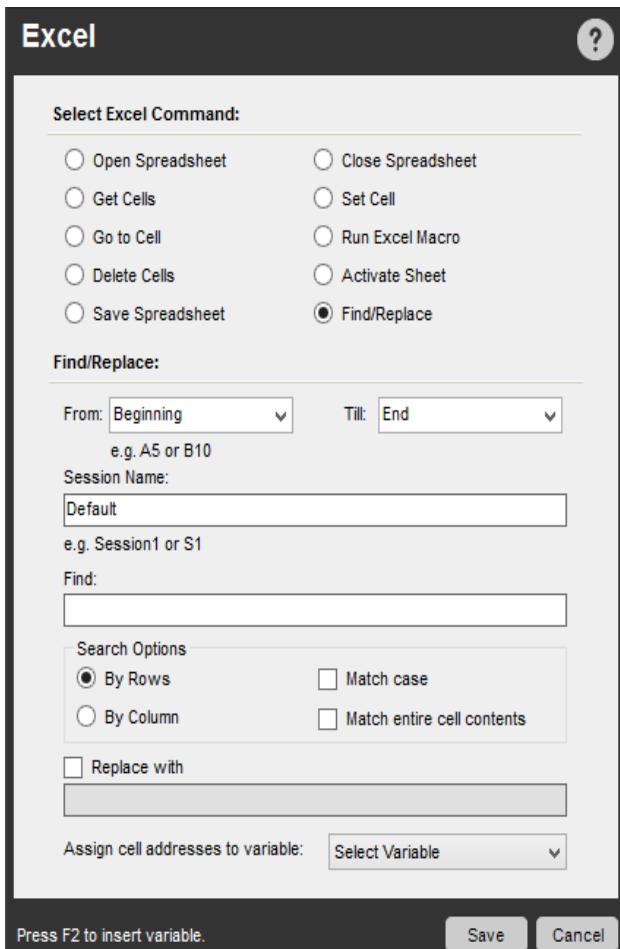
Save Spreadsheet Command

Use this command to save a spreadsheet in an open workbook.

(Image as seen in Enterprise Edition 8.2)

Find/Replace Command

Use this command to find and replace data in a spreadsheet within an open workbook. You can specify a cell range with specific search options.

(Image as seen in Enterprise Edition 8.2

Using the Load Add-ins Option

Use this option when opening an Excel spreadsheet in which Add-ins are required.

Using the Sub-Commands

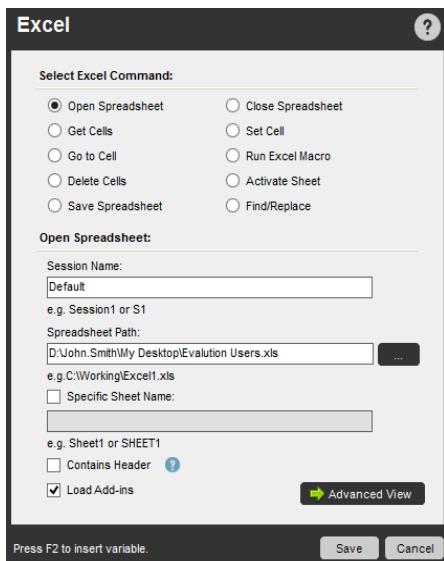To insert an Excel command in your task, follow these steps:

1. Double-click or drag one of the **Excel** commands to the Task Actions List pane. The Excel window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.

Use Case Scenario

Copying data from Excel to Notepad -

To copy the content (data) of selected cells from Excel to notepad, follow the steps given below:
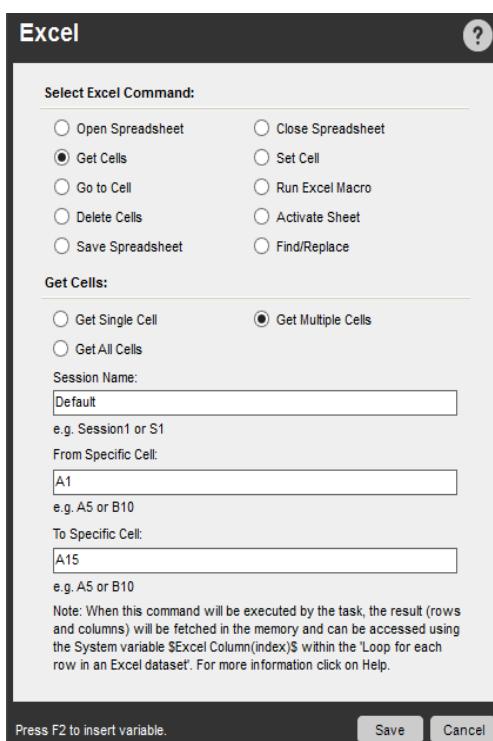
1. Launch the Task Editor.

2. Drag and drop or double click the 'Open Spreadsheet' option under the 'Excel Command' node to the Task Actions List pane.

(Image as seen in Enterprise Edition 8.2)

   a. Specify the session name and spreadsheet path. Select the option 'Specific Sheet Name' if it has more than one spread-sheet worth of data.

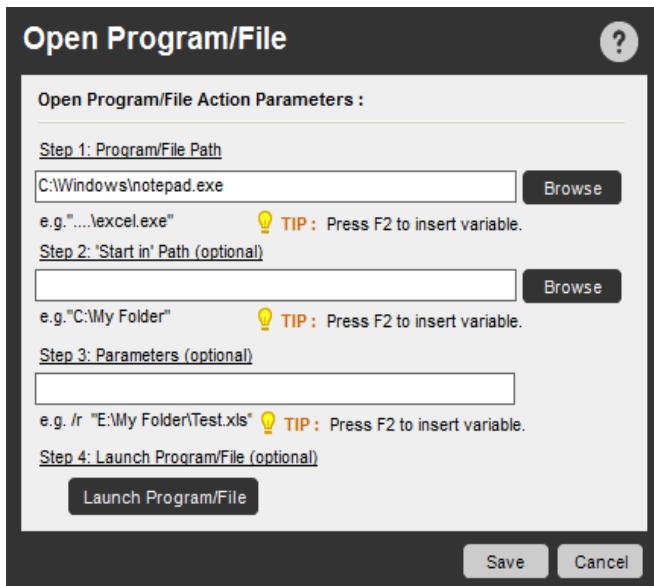   b. Also select the 'Load Add-in' option, if applicable.

   c. Click Save.

3. Drag and drop/double click the 'Get Cell' option under Excel Command to copy the cell data.
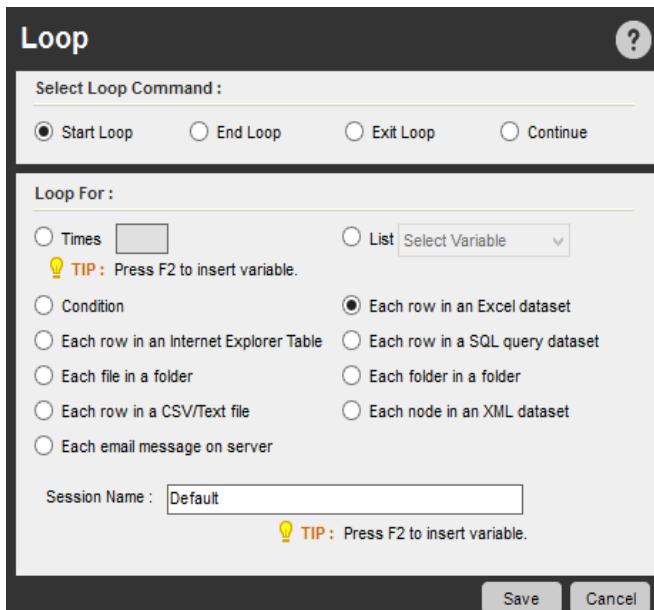

(Image as seen in Enterprise Edition 8.2)

   a. Specify the option 'Get Multiple Cells', session name and cell range in the 'From Specific Cell' & 'To Specific Cell' to copy multiple cells from the spreadsheet.

b. Click save.

• The next step involves opening the application to which the cells are to be copied.

5. Drag and drop/double click the 'Open File/Folder' command.

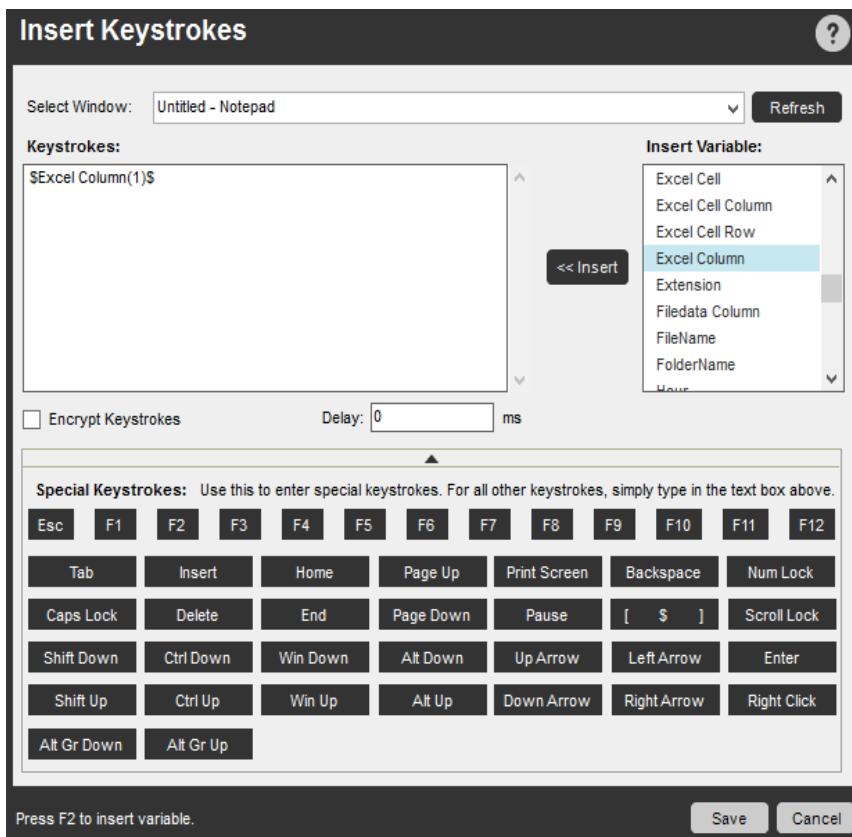*(Image as seen in Enterprise Edition 8.0)*

a. Specify the file name or select from a specific path by clicking [Browse] .

b. You can provide the 'Start in' Path or set 'Parameters' and 'Launch Program/File' optionally.

c. Click save.

• At this juncture, you need to insert a loop that allows you to copy each row from the spreadsheet specified in the 'Open Spread-sheet' session.

6. Drag and drop/double click the 'Loop' command.
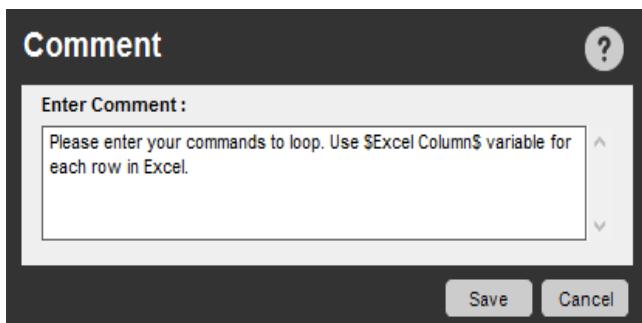

(Image as seen in Enterprise Edition 8.2)

a. Select 'Each row in an Excel dataset' and save.

• Now assign the copied data to a variable.

7. Drag and drop/double click the 'Insert Keystroke' Command.

Insert Keystrokes

Select Window:	Untitled – Notepad ⌄ Refresh

Keystrokes:

$Excel Column(1)$

Insert Variable:

- Excel Cell
- Excel Cell Column
- Excel Cell Row
- Excel Column
- Extension
- Filedata Column
- FileName
- FolderName
- Hour

<< Insert

☐ Encrypt Keystrokes   Delay: 0   ms

▲

Special Keystrokes: Use this to enter special keystrokes. For all other keystrokes, simply type in the text box above.

Esc	F1	F2	F3	F4	F5	F6	F7	F8	F9	F10	F11	F12

Tab	Insert	Home	Page Up	Print Screen	Backspace	Num Lock
Caps Lock	Delete	End	Page Down	Pause	[$]	Scroll Lock
Shift Down	Ctrl Down	Win Down	Alt Down	Up Arrow	Left Arrow	Enter
Shift Up	Ctrl Up	Win Up	Alt Up	Down Arrow	Right Arrow	Right Click
Alt Gr Down	Alt Gr Up					

Press F2 to insert variable.   Save   Cancel

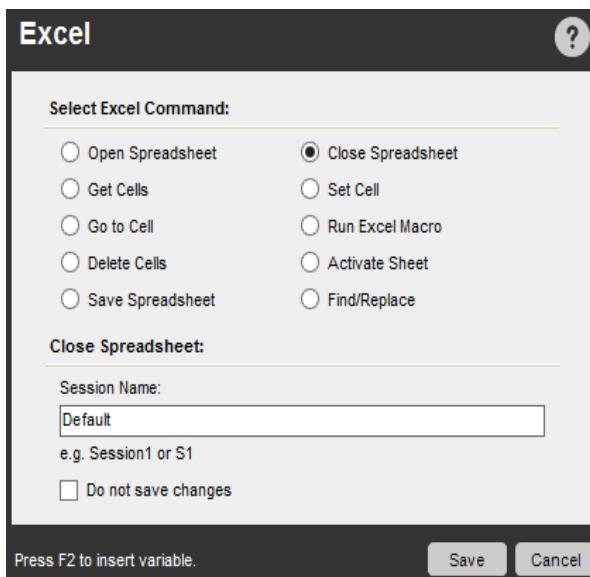*(Image as seen in Enterprise Edition 8.2)*

    a. Select the file to which the data is to be copied from the 'Select Window' drop down.

    b. Select 'Encrypt Keystrokes' to enable encryption.

    c. Insert appropriate keystrokes and variables and click save.

8. Include the following comment and save:

Comment

Enter Comment :

Please enter your commands to loop. Use $Excel Column$ variable for each row in Excel.

Save   Cancel

(Image as seen in Enterprise Edition 8.0)

📝 Note: This will mark end of the loop.

9. Drag and drop/double click the 'Close Spreadsheet' option:

(Image as seen in Enterprise Edition 8.2)

   a. Specify the 'Session Name' and save.

- The following depicts the above created task:

10. Save the task.

Learn more about similar examples from our online section Excel Automation

---

Also See...

- Excel Command - Examples
- App Integration Command
- Database Command
- Email Automation Command
- Loop Command
- OCR Command
- PDF Integration Command
- Read from CSV-Text Command
- Terminal Emulator Command
- XML Command

Excel Command - Contains Header

Applicable from version 8.1.2 onward

When using Excel operations for fetching/retrieving large amounts of data from excel spreadsheets that has its first row defined as a heading, you need to specify the rows and columns in a manner that they take those headings into consideration.

The 'Contains Header' option allows you to configure the command parameters to include or exclude headers; as the case may be.
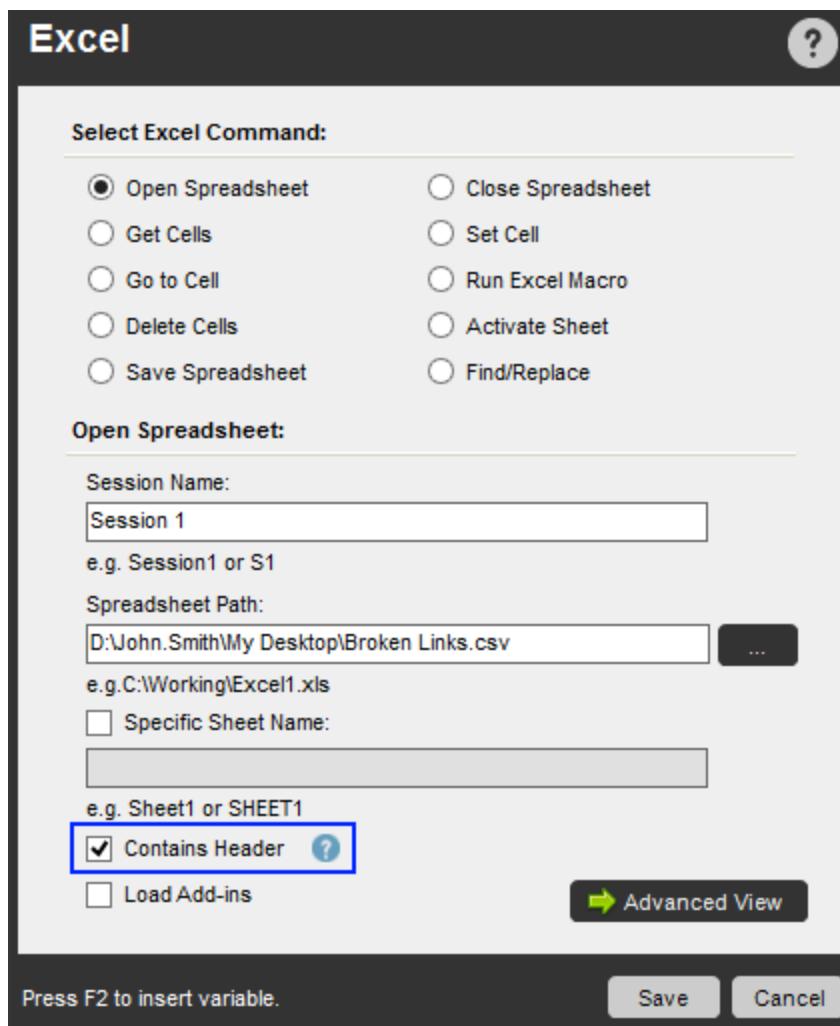
- A sample Excel spreadsheet with Headings:

	A	B	C
1	**ID**	**Name**	**Designation**
2	123	John Smith	Product Manager
3	124	Dwane Williams	Support Lead
4	125	Jeremy McCullum	Software Engineer

Contains Headers in Excel operations

Contains Header allows you to retrieve/fetch data from the active spreadsheet that has its first row defined as a header. You can **define** the option in **Open Spreadsheet** for a **particular session**. It is applicable to Excel operations Get Cells, Set Cell, Go to Cell, Find/Replace and Delete Cell for that particular session (as configured in Open Spreadsheet).

Also, in cases where one or more columns of the target Excel spreadsheet change position due to any update (i.e. introducing new columns/deleting existing columns), the corresponding automation task has to be updated to accommodate the change. Using 'Contains Header' you can enable the task to automatically map re-positioned columns of the target Excel spreadsheet.

Excel Cell Variable - a must for Contains Header option

You can use the System Variable - Excel Cell when configuring Contains Header. This variable is **exclusive** to Contains Header.

Excel			(This category contains Excel related variables.)
Excel Cell	Excel Cell	String	Use in Excel operation like Set Cell, Delete Cells, Go to Cell when contains header option is set to indicate column name and position of the cell from the header or in the column value.
Excel Cell Column	Excel Cell Column	Column	Returns the column of the Excel on which the operation has been performed.
Excel Cell Row	Excel Cell Row	Row	Returns the row of the Excel on which the operation has been performed.

Excel Cell should be used while configuring Excel commands Get Cells, Set Cell, Go to Cell, Find/Replace and Delete Cell. During play time, the variable will indicate the column name and position of the cell from the header. It gives the column reference to the defined cell; e.g cell defined is A5, it will refer to **Row A , Column 6**, if 'Contains Header' is enabled.

> Note: The string defined in the variable has to be an exact match. For instance, while retrieving/fetching data from each cell/specified cells under the heading 'Id', use the System variable $Excel Cell(Id,2)$; variables such as '$Excel Cell(id,2)$' or '$Excel Cell(<space>Id<space>),2$' will be considered invalid.

When you use this variable, without configuring the Open Spreadsheet command, the program throws an error during play time - "To enable Excel Cell variable configure Open Spreadsheet command."

To know more about the variable, refer Excel System Variables.

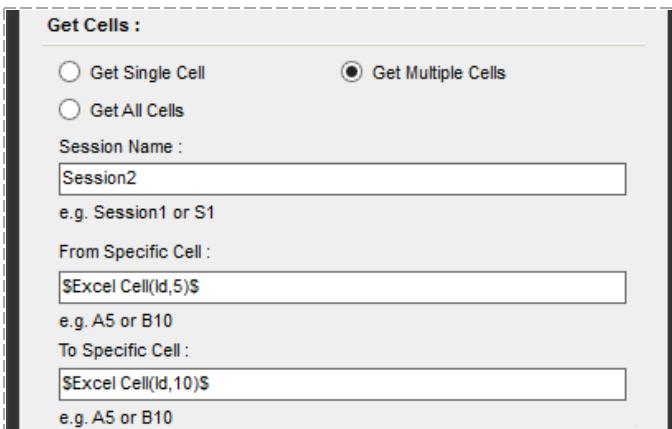## The where and how of Contains Header

You can apply the Contains Header option for Excel operations:

1. **Open Spreadsheet -** Contains Header option is controlled from this sub-command as it is configured to the session specified here. Consequent Excel operations will consider the option; whether selected or not for the session specified in this command. Hence, configuring this command is a must.

   Reference Points:
   - When you open multiple spreadsheets during automation, the program considers the last spreadsheet action. It means that if 'Contains Header' is enabled in Session 1 and not in Session 2, the consequent command **will not** have Contains Header enabled **unless you specify** the particular session that has Contains Header (Session 1 in this case).
   - In spreadsheets with duplicate header name/content, the program will consider the first occurrence of the header name/content.
   - If you include an invalid value or the header row mismatches, the program will throw an error during play time.
   - You can input maximum three (3) alphabets for a Column title (e.g. ABC); if you input more then you will encounter an error "Check the value specified in the Column Title. You can input up to 3 alphabets only."

2. **Get Cells -** For the 'Get Single Cell' or ' Get Multiple Cells' parameters, include the Excel Cell variable in the 'Specific Cell' or 'From Specific Cell' and 'To Specific Cell' text boxes as shown:

In the variable, **input the cell heading title**. For e.g. if the cell range specified is A5 through A10, define the variable with

heading title as given in cell A1; in this case "ld".

> 📝 Note: In editions earlier to 8.1, Contains Header was applicable and available in Get Cells command only. If you are re-configuring a task that uses that version with 'Contains Header' enabled, it will be "grayed out." It is recommended that you re-configure the command in the task if you want the task to run properly in the current edition.

- **Set Cell -** You can apply the 'Contains Header' option to parameters Active Cell and Specific Cell when configuring the Set Cell command. If the cell position shifts, values in the specified cells will be set appropriately.

  You cannot, however, reset value in the cells if the selected cell, is a header cell. You will encounter the error "*The specified cell is part of the header row. You cannot set value in it as 'Contains Header' is enabled*." during play time.

  You will also not be allowed to set the value of a cell that is defined as a header title. Same is true, if it's in edit mode and the option is enabled.

- **Go to Cell** - You can Go to a specific or active cell when 'Contains Header' is enabled. All conditions that are applicable during Set Cell are also applicable here.

> 📝 Note: In Automation Anywhere Excel Command, **Active Cell > End of the row/column** refers to "end of the row/column of data"; **not** the end of spreadsheet row/column.

- **Delete Cells** - When using this command for 'Contains Header', you will **not be** allowed to delete the cells that have a header title defined i.e. the first row of the spreadsheet. If you have specified a cell that is defined as a header - either by using the active or specific cell parameter, you will encounter the error "*The specified cell is part of a header row. You cannot delete it as 'Contains Header' is enabled*." during play time.

- **Find/Replace -** When using this command for 'Contains Header', you will be allowed to **Find** a cell value by Row/Column but **Replace** the content in **only** those cells that are **not** defined as header titles. If you specify a cell that is defined as a header, you will encounter the error "*The specified cell is part of a header row. You cannot replace its value as 'Contains Header' is enabled*." during play time.

Also See...

- Excel Command
- Excel Command - Examples
- Excel System Variables

File and Folder Commands

Use the File and Folder commands to automate common operations that involve manipulating files and folders.

Overview

These days, most occupations require the organizing and manipulation of files and folders on computer systems. Automating these tedious tasks can help organizations become more efficient. In addition, less human error occurs when automating.
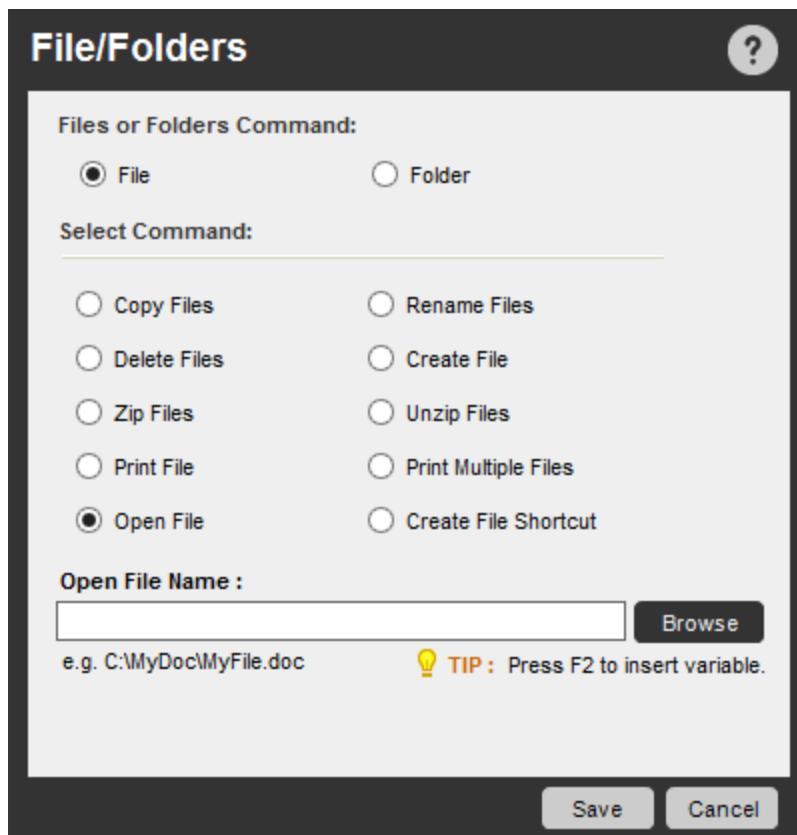
The File and Folder commands provide an easy way to automate creating, opening, copying, moving, renaming, deleting, and organizing files and folders.

The File and Folder commands also offer advanced operations for working with files and folders based on their size and date created or modified.

Using the Sub-Commands
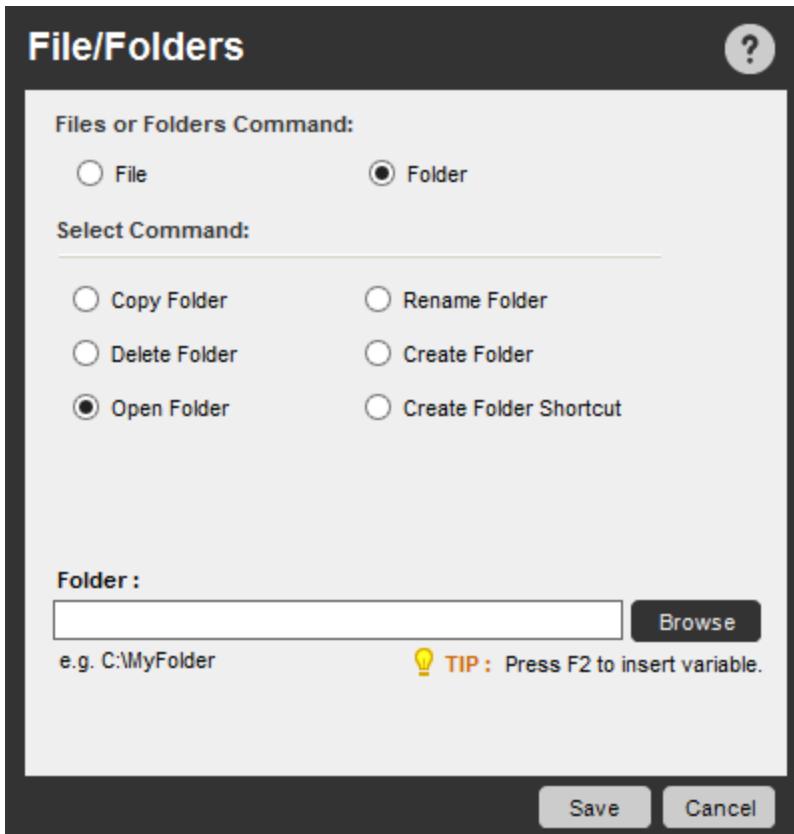
Open File Command

Use this command to open an existing file.
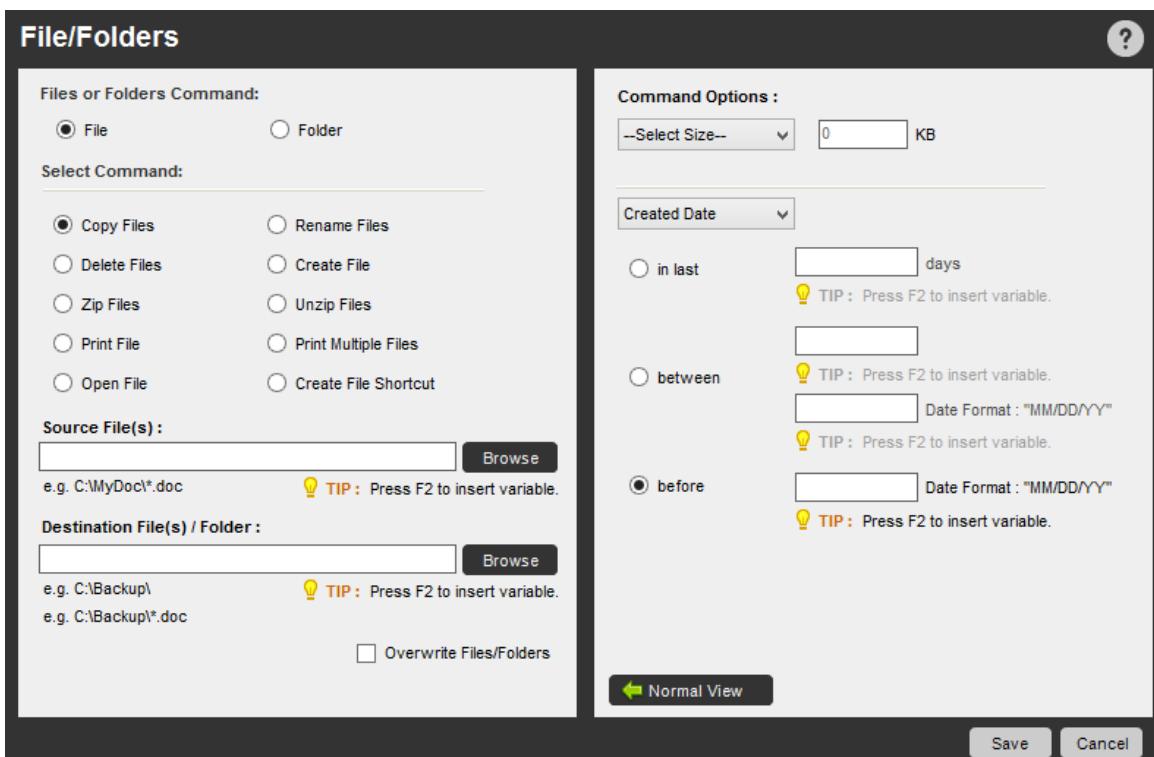
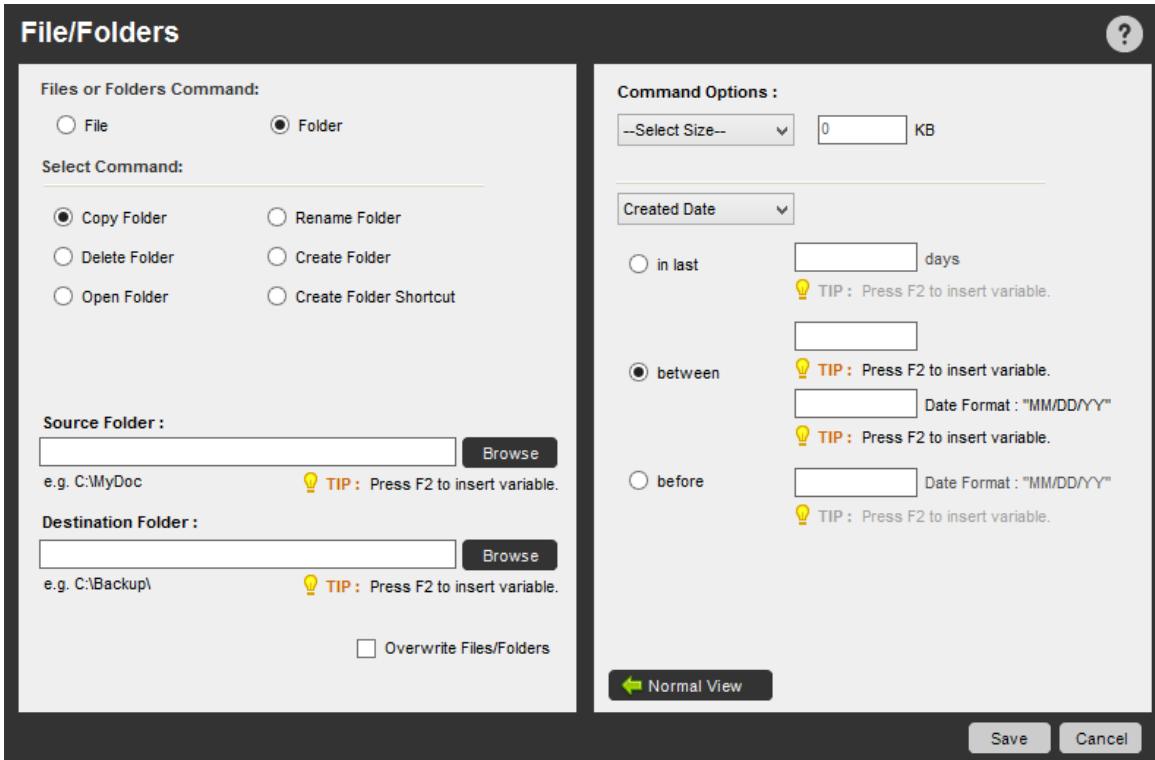## Open Folder Command

Use this command to open an existing folder.

Copy Files Command

Use this command to copy an existing file.

Copy Folder Command

Use this command to copy an existing folder.

Rename Files Command

Use this command to rename an existing file.

Rename Folder Command

Use this command to rename an existing folder.

Delete Files Command

Use this command to delete an existing file.

Delete Folder Command

Use this command to delete an existing folder.

Create File Command

Use this command to create a new file.
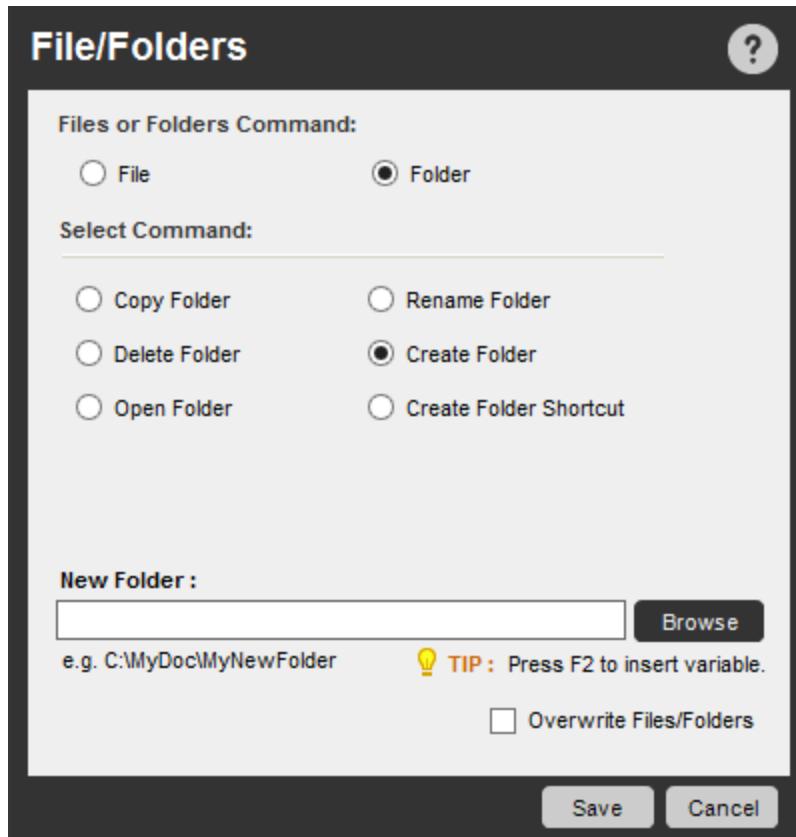
Create Folder Command

Use this command to create a new folder.
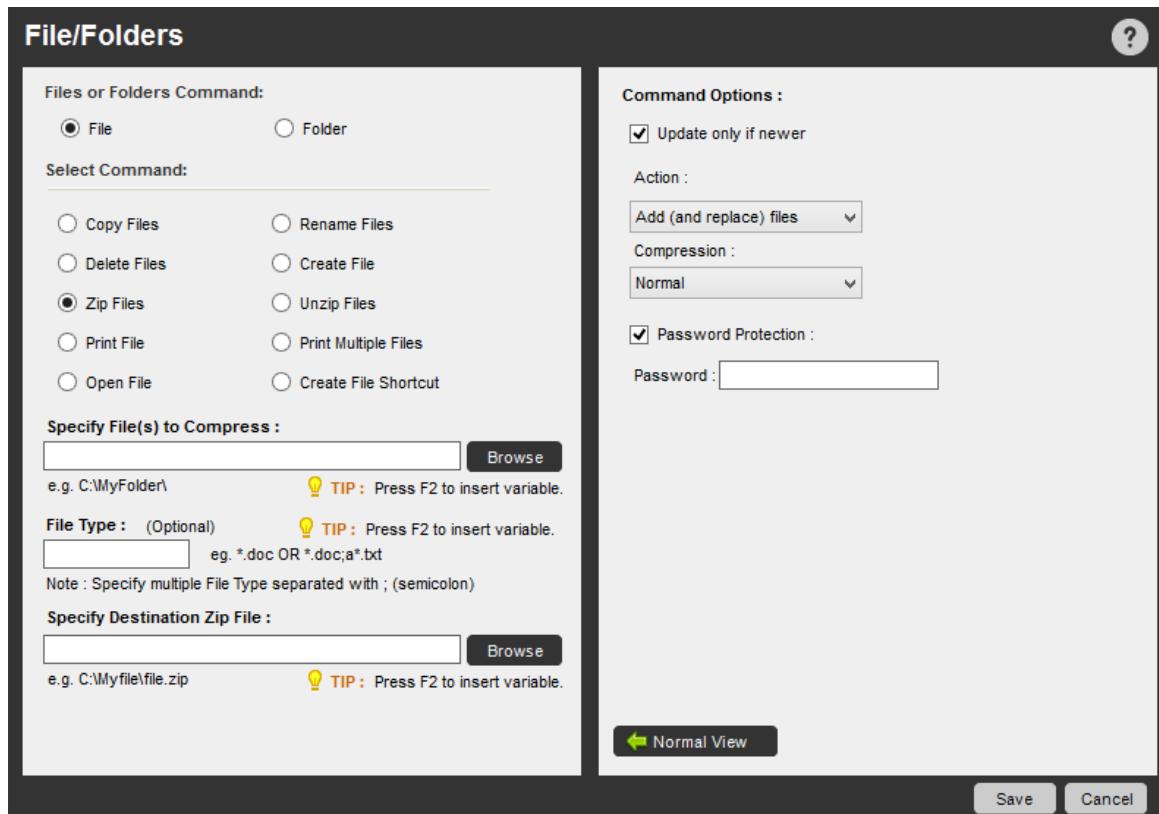
Zip Files Command

Use this command to compress a group of files into a ZIP file.
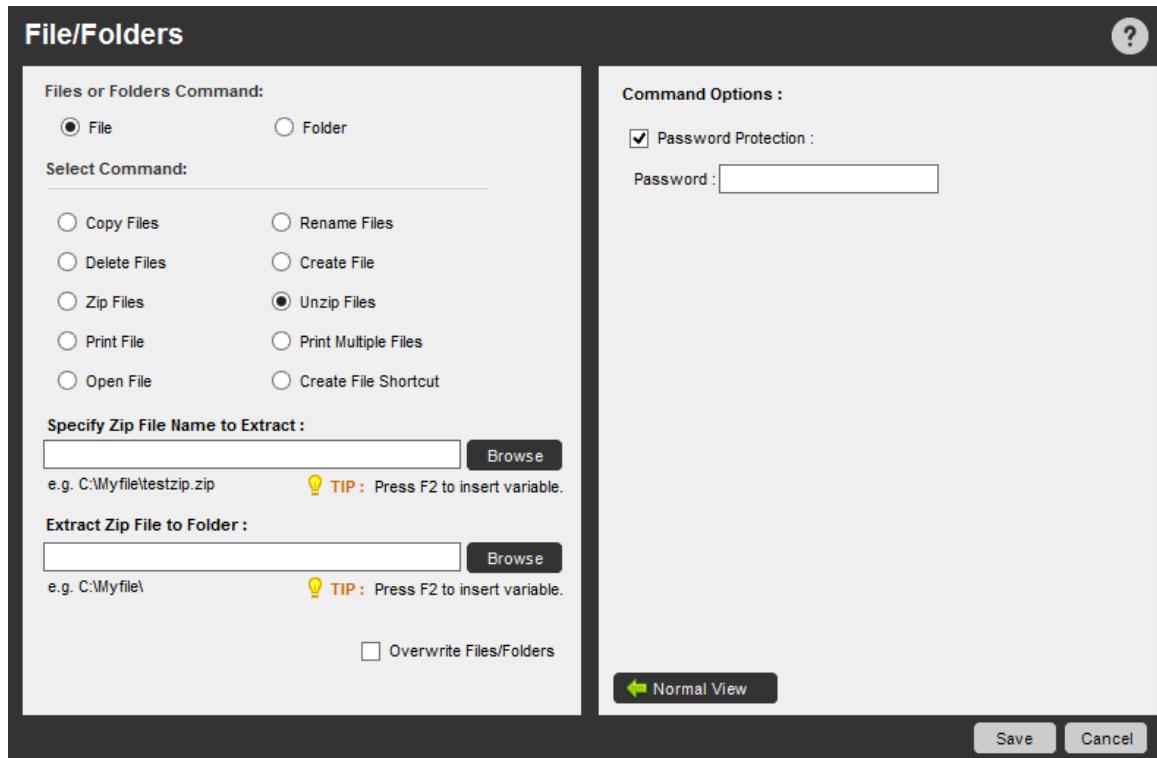
Unzip Files Command

Use this command to unzip and extract a group of files that has previously been zipped.
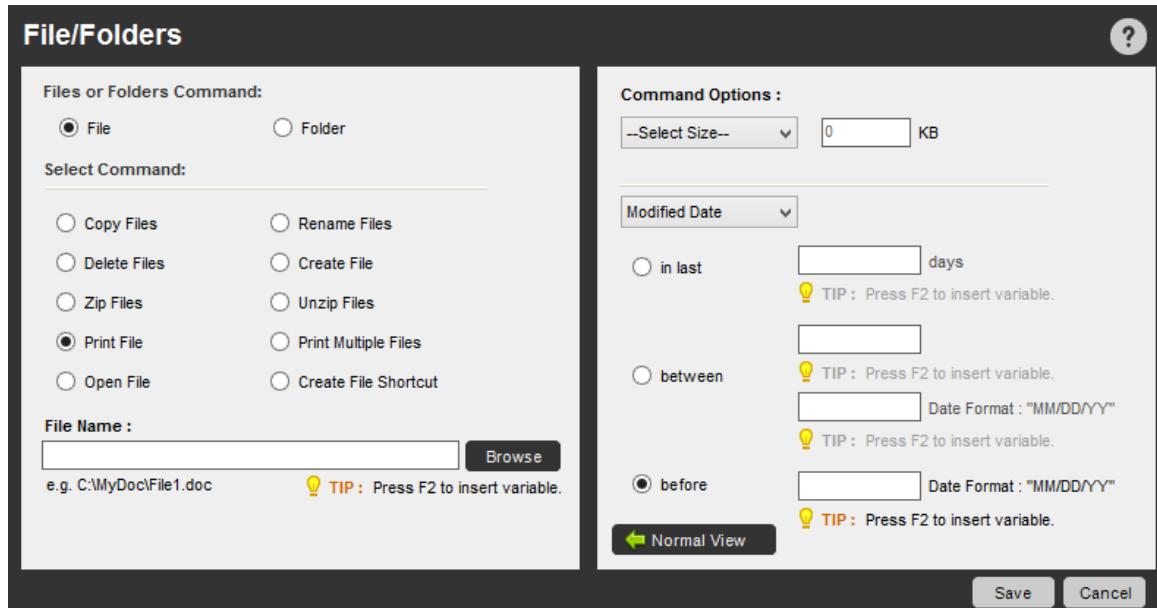
Print File Command

Use this command to print a file.

Other User
Resources:

User forums

Ask the Expert

Open a support case

Automation Examples
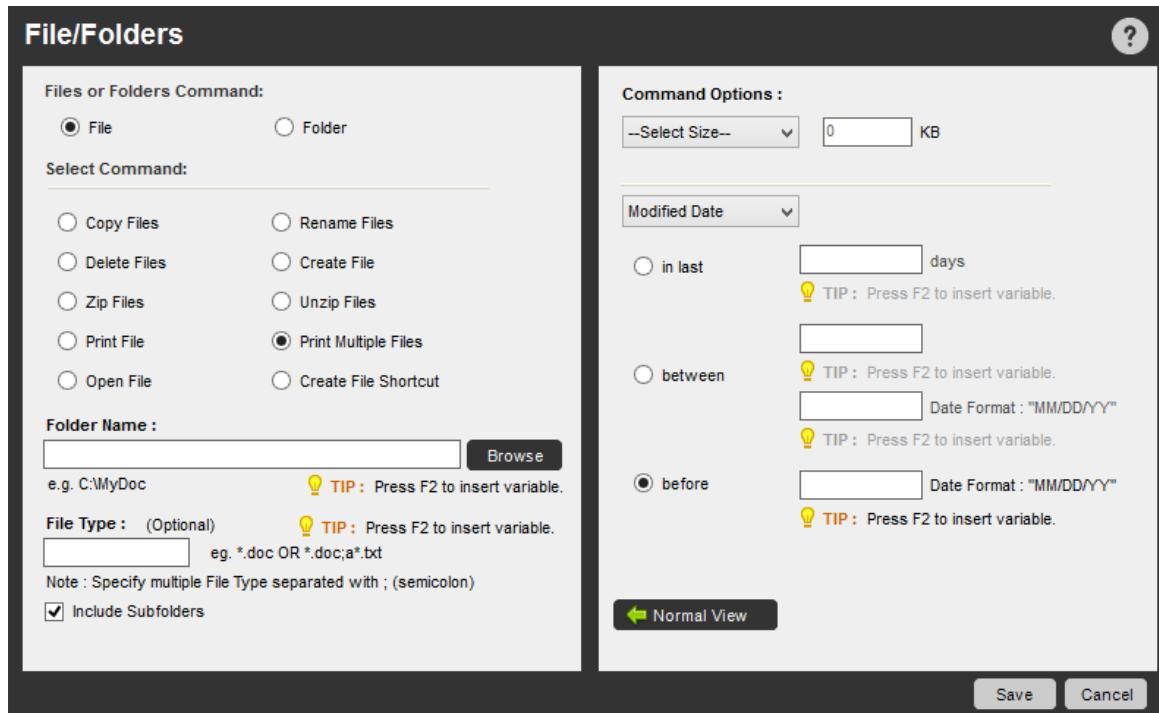
Print Multiple Files in a Folder Command

Use this command to print all of the files that reside in a folder.
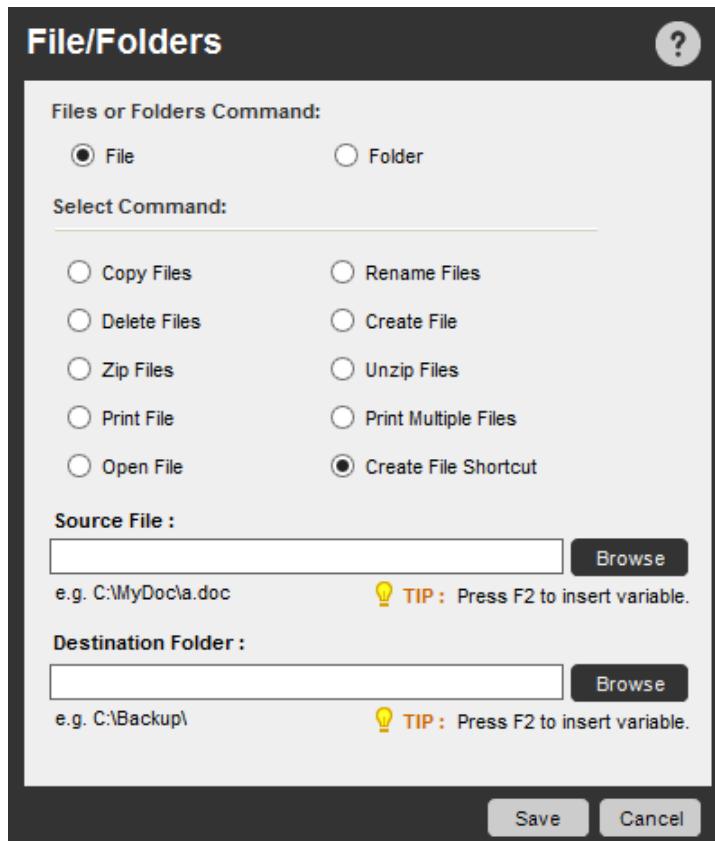
File/Folders

Files or Folders Command:
- ( ) File
- ( ) Folder

Select Command:
- ( ) Copy Files
- ( ) Rename Files
- ( ) Delete Files
- ( ) Create File
- ( ) Zip Files
- ( ) Unzip Files
- ( ) Print File
- (•) Print Multiple Files
- ( ) Open File
- ( ) Create File Shortcut

Folder Name :

[   ] [Browse]

e.g. C:\MyDoc   💡 TIP : Press F2 to insert variable.

File Type : (Optional)   💡 TIP : Press F2 to insert variable.

[   ]   eg. *.doc OR *.doc;a*.txt

Note : Specify multiple File Type separated with ; (semicolon)

[✓] Include Subfolders

Command Options :

--Select Size-- [v]   [0] KB

Modified Date [v]

- ( ) in last   [   ] days
                💡 TIP : Press F2 to insert variable.
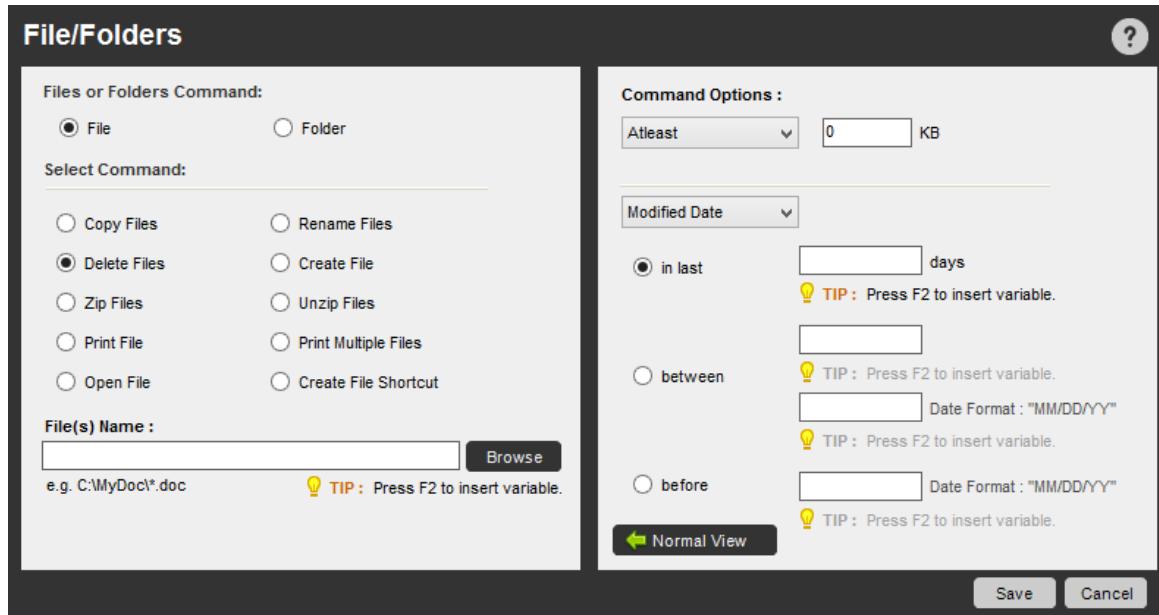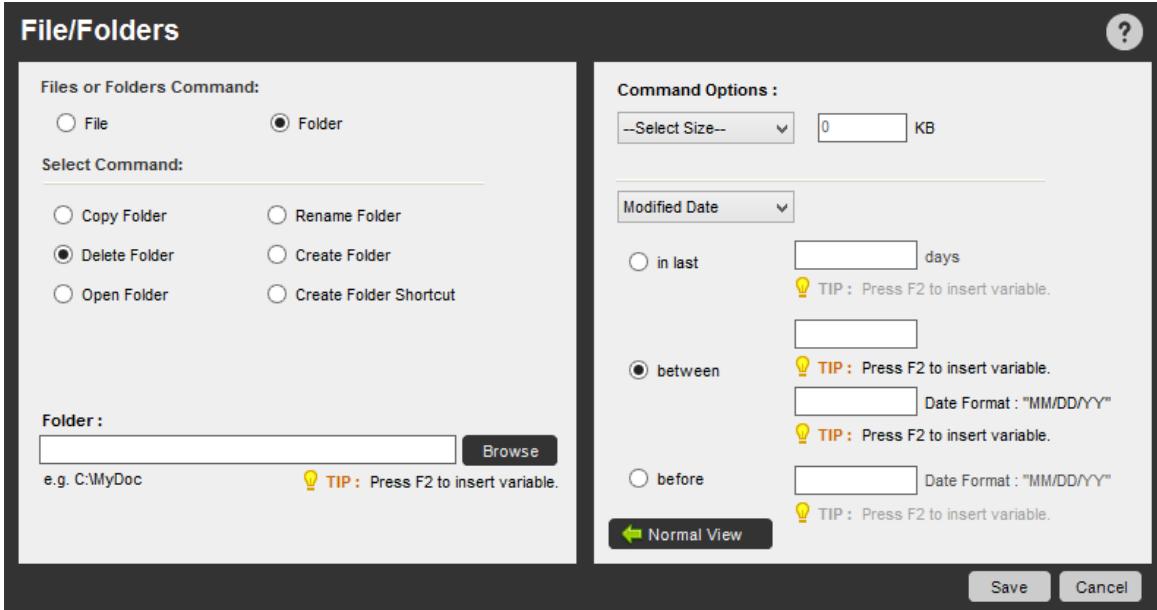
- ( ) between   [   ]
                💡 TIP : Press F2 to insert variable.
                [   ] Date Format : "MM/DD/YY"
                💡 TIP : Press F2 to insert variable.

- (•) before   [   ] Date Format : "MM/DD/YY"
                💡 TIP : Press F2 to insert variable.

[← Normal View]

[Save] [Cancel]

Create File Shortcut Command

Use this command to create a shortcut for opening a file. For example, you might want to create a shortcut to a file on a user's desktop.
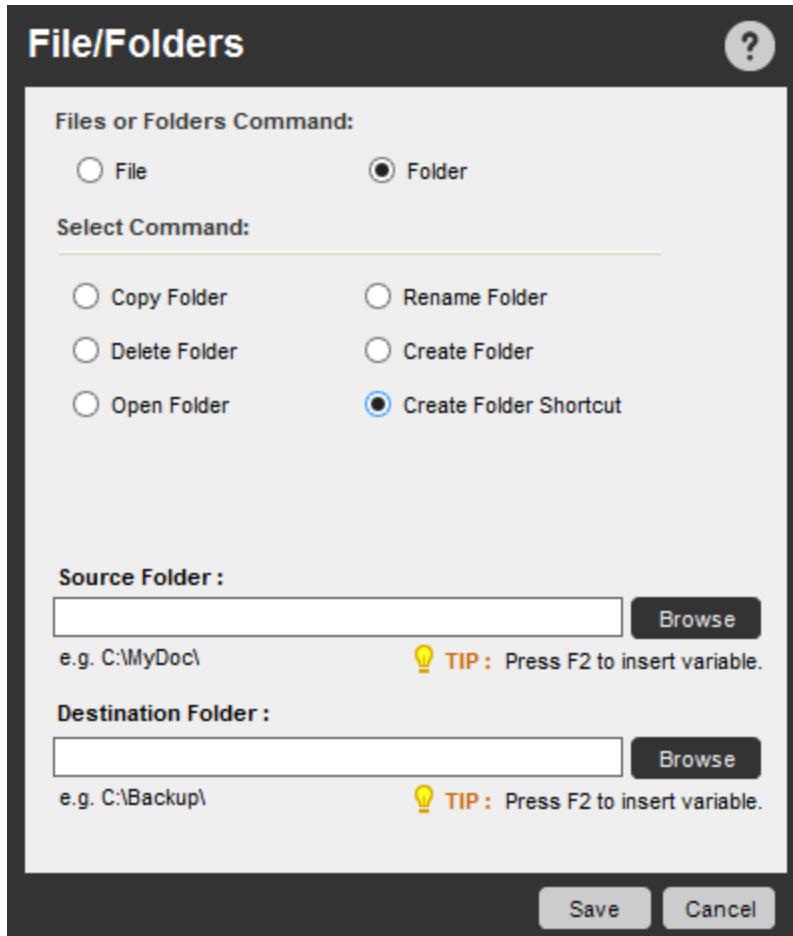
File/Folders

Files or Folders Command:
- (•) File
- ( ) Folder

Select Command:
- ( ) Copy Files
- ( ) Rename Files
- ( ) Delete Files
- ( ) Create File
- ( ) Zip Files
- ( ) Unzip Files
- ( ) Print File
- ( ) Print Multiple Files
- ( ) Open File
- (•) Create File Shortcut

Source File :

[   ] [Browse]

e.g. C:\MyDoc\a.doc   💡 TIP : Press F2 to insert variable.

Destination Folder :

[   ] [Browse]

e.g. C:\Backup\   💡 TIP : Press F2 to insert variable.

[Save] [Cancel]

Create Folder Shortcut Command

Use this command to create a shortcut for opening a folder. For example, you might want to create a shortcut to a folder on a user's desktop.

Also See...

- File/Folders Command - Examples
- Log-to-File Command
- Manage Windows Controls Command
- Open Program/File Command
- System Command
- Windows Actions Command

FTP/SFTP Command

Use the Automation Anywhere FTP/SFTP command to automate any FTP task.

Overview

FTP (File Transfer Protocol) is a standard network protocol used to transfer files from one host to another host over a TCP-based network, such as the Internet.

You can use the Automation Anywhere FTP/SFTP command to automate the following:

- Use FTP for file uploads and downloads
- Back up websites
- Use Secure FTP for secure remote file transfer

The FTP/SFTP command provides an Advanced View option for working with files and folders based on when they are created or modified.

Using the FTP/SFTP Command

To insert an FTP/SFTP command in your task, follow these steps:

1. Double-click or drag one of the **FTP/SFTP** commands to the Task Actions List pane. The FTP window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.

The FTP/SFTP command includes the following options:

- **Connect** to FTP/SFTP server: Specify the FTP/SFTP server name, user name, and password. Anonymous login to FTP/SFTP servers is supported. Click the Advanced View button to set advanced connection options.
- **Disconnect** from FTP/SFTP server: Terminates the connection to an FTP/SFTP server.
- **Put Files:** Uploads one or more files from a folder.
- **Get Files:** Downloads one or more files to a local machine from an FTP/SFTP folder.

  The FTP/SFTP command also offers advanced operations for working with files based on the date they are created or modified.

- **Put Folder:** Upload an entire folder from a local machine to an FTP/SFTP server. You can specify filters using wild card characters to restrict uploads to specific types of files.
- **Get Folder:** Download an entire folder from an FTP/SFTP server to a local machine. You can specify filters using wild card characters to download only specific types of files.

  The FTP/SFTP command also offers advanced operations for working with folders based on the date they are created or modified.

- **Delete Files:** Delete one or more files from an FTP/SFTP folder.
- **Rename Files:** Rename one or more files in an FTP/SFTP folder.
- **Create Folder:** Create a folder on an FTP/SFTP server.
- **Delete Folder:** Delete a folder (including all sub-folders and files within it) from an FTP/SFTP server.
- **Change Folder:** Move to a different folder on an FTP/SFTP server. Use this option when you want to download or upload files

in more than one directory on your FTP/SFTP server.

Also See...

- Internet Connection Command
- Launch Website Command
- Send Email Command
- Web Recorder Command
- Web Service Command

IF/ELSE Command

Use this command to add conditional logic and actions to your automated task.

Sub-Commands

The IF/ELSE command provides the following sub-commands:

- File Exists / File Does Not Exist
- Folder Exists / Folder Does Not Exist
- Window Exists / Window Does Not Exist
- Application Running / Application Not Running
- Script Successful / Script Unsuccessful
- Task Successful / Task Unsuccessful
- Ping Successful / Ping Unsuccessful
- Variable
- File Size
- File Date
- Web Control
- Image Recognition
- Service Running / Service Not Running
- Windows Control
- Else If
- Else
- End If

Overview

One of the most powerful features of Automation Anywhere is the IF/ELSE command. You can use the sub-commands to perform actions when certain conditions exist.

When using most of the conditional sub-commands, you can specify how long to wait for the condition to be true before taking another action.
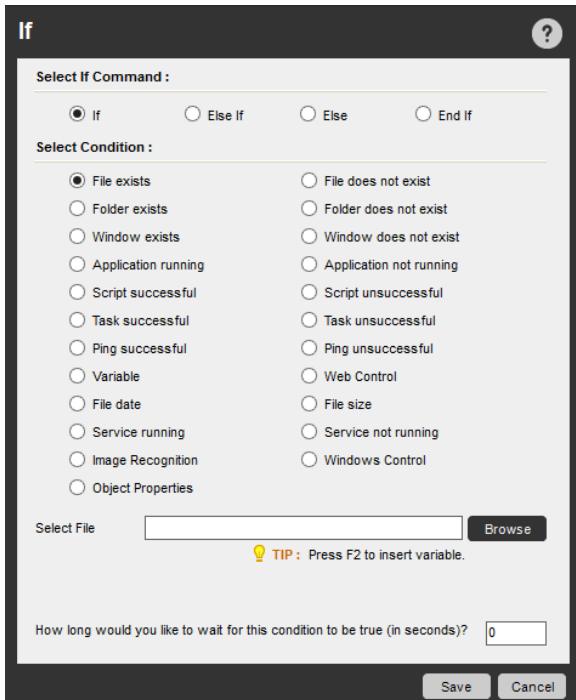
Using the Sub-Commands

To insert an IF/ELSE command in your task, follow these steps:

1. Double-click or drag one of the **IF/ELSE** commands to the Task Actions List pane. The IF/ELSE window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.
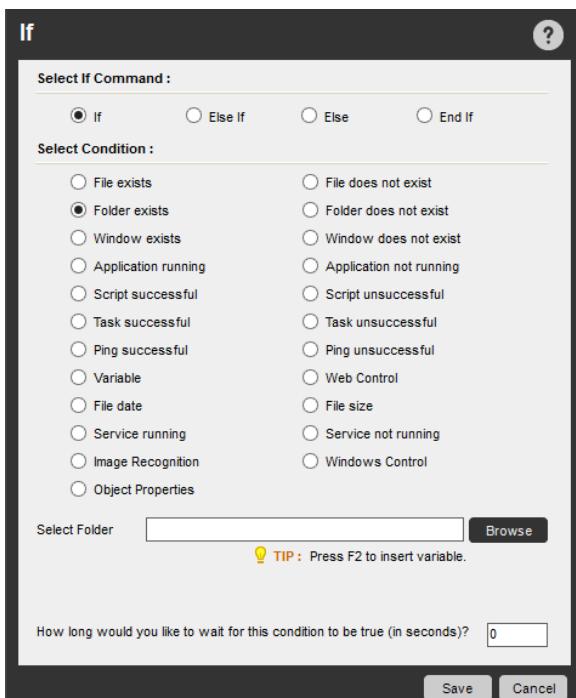
File Exists/File Does Not Exist Commands

Use these commands when you want to perform an action based on whether a file exists. For example, if a data file exists, you might format the file and upload it to a database.

Folder Exists/Folder Does Not Exist Commands
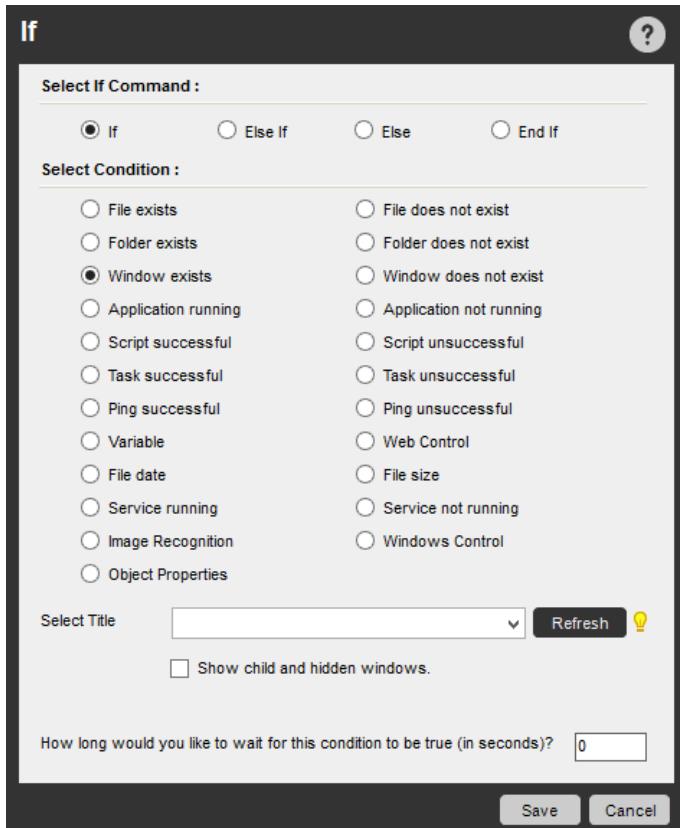
Use these commands when you want to perform an action based on whether a folder exists.

Window Exists/Window Does Not Exist Commands

Use these commands to do the following:
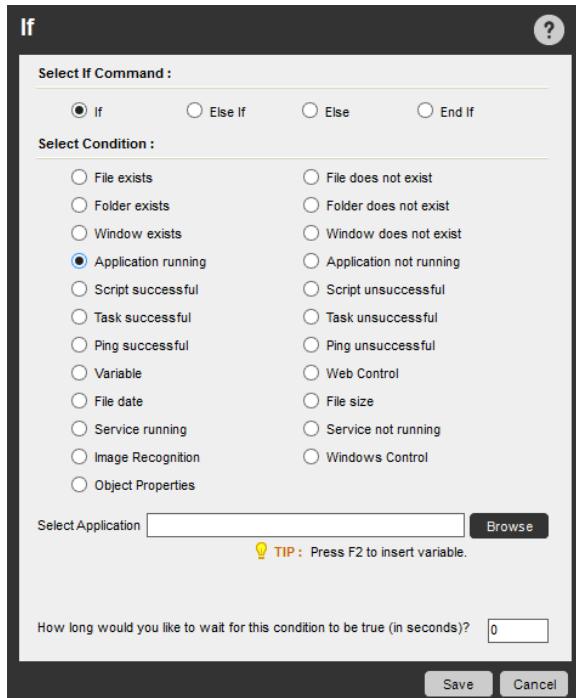
- Check or verify that a specific application is open.
- Check whether an error has occurred.
- Check whether a file download has completed.

These are some of the most frequently used commands in Automation Anywhere.

Application Running/Application Not Running Commands

Use these commands to perform an action based on whether an application is running. When a program is running in the background, you can still perform actions by using this command.

Script Successful/Script Unsuccessful Commands

Use these commands to integrate third-party scripts with your automation task. After the script runs and issues a return code, Automation Anywhere can determine if the script ran successfully.
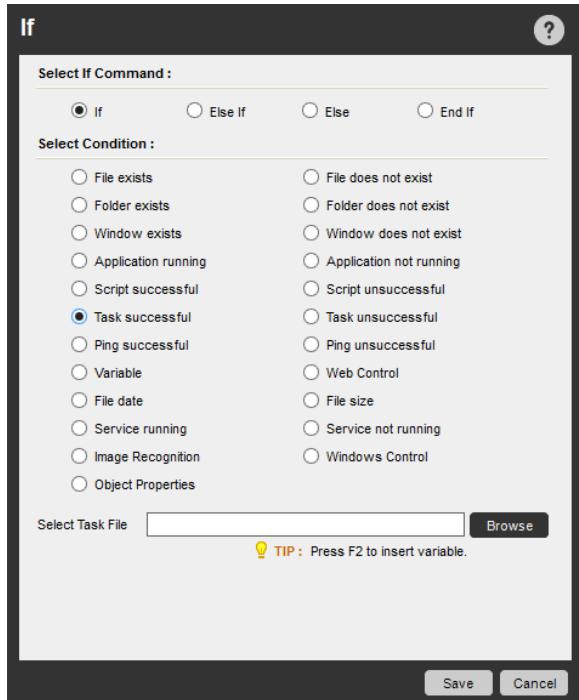
For scripts that you have already developed, you can use this command to integrate them with your automation task.

Task Successful/Task Unsuccessful Commands

Use these commands to run several Automation Anywhere tasks, one after another. The command runs the selected task, and depending on the return code, determines if it ran successfully and takes appropriate actions based on the result. For example, if a task that monitors your website runs successfully, it means that your website is up and no action is required. However, if the task is unsuccessful, it means that the website is down and you need to take action.

If (?)

Select If Command :

◉ If ○ Else If ○ Else ○ End If

Select Condition :

○ File exists ○ File does not exist
○ Folder exists ○ Folder does not exist
○ Window exists ○ Window does not exist
○ Application running ○ Application not running
○ Script successful ○ Script unsuccessful
◉ Task successful ○ Task unsuccessful
○ Ping successful ○ Ping unsuccessful
○ Variable ○ Web Control
○ File date ○ File size
○ Service running ○ Service not running
○ Image Recognition ○ Windows Control
○ Object Properties

Select Task File [   ] [Browse]

💡 **TIP :** Press F2 to insert variable.

[Save] [Cancel]

Ping successful/Ping Unsuccessful Commands

Use these commands to check whether a machine or server is running, and take appropriate action based on the result.

If (?)

Select If Command :

◉ If ○ Else If ○ Else ○ End If

Select Condition :

○ File exists ○ File does not exist
○ Folder exists ○ Folder does not exist
○ Window exists ○ Window does not exist
○ Application running ○ Application not running
○ Script successful ○ Script unsuccessful
○ Task successful ○ Task unsuccessful
◉ Ping successful ○ Ping unsuccessful
○ Variable ○ Web Control
○ File date ○ File size
○ Service running ○ Service not running
○ Image Recognition ○ Windows Control
○ Object Properties

IP Address / Machine Name [   ]

💡 **TIP :** Press F2 to insert variable.

How long would you like to wait for this condition to be true (in seconds)? [0]

[Save] [Cancel]

Variable Command

This command, though it looks simple, is perhaps the most powerful conditional command. Use this command to perform hundreds of checks.
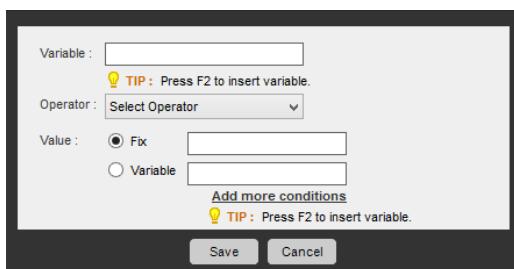
For example, you can copy data from an application or website and verify that the data you copied matches a specific text string. You can also verify a numeric string, or check to see if it is greater than or less than a variable or a fixed value.

Using AND or OR conditions in IF Variable command*
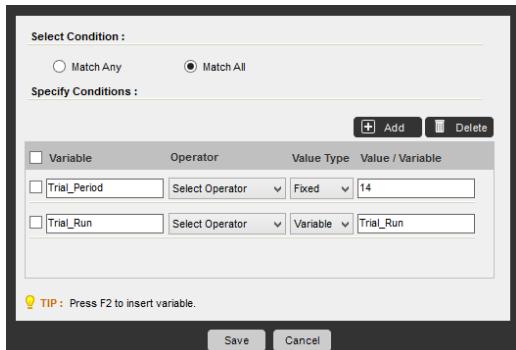
* Available from version 7.5.2 onward

When you need to include the 'AND' or 'OR' conditions in conjunction with the 'Variable' sub-command, use the 'Add more conditions' option. It offers you the luxury of including all your AND or OR conditions within a single parameter, instead of specifying separate parameters for each condition as was the case in earlier editions of Automation Anywhere.

(Image as seen in Enterprise Edition 8.0)

Enable the 'Match Any' option for 'OR' conditions or 'Match All' for 'AND' conditions (refer image below)

(Image as seen in Enterprise Edition 8.0)
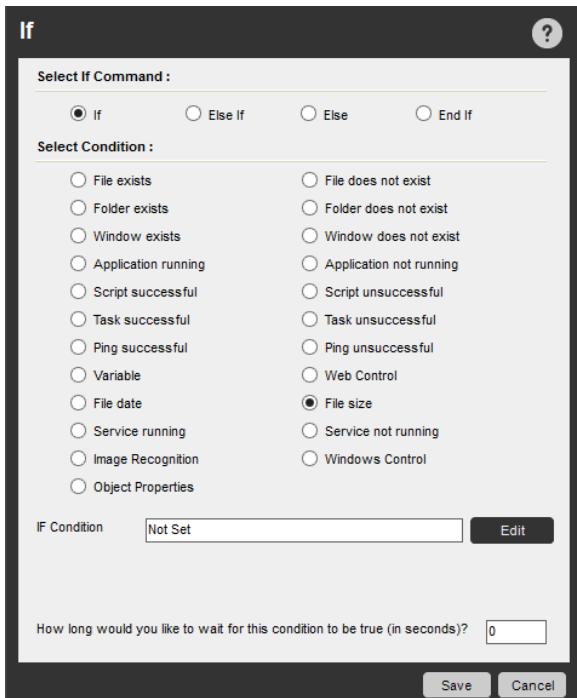
> 📝 Note: You cannot include both; 'Match Any' + 'Match All' within a parameter. You are required to specify them within a task, separately.

Reference the examples topic to understand how to shorten your tasks when using 'AND' or 'OR' conditions.
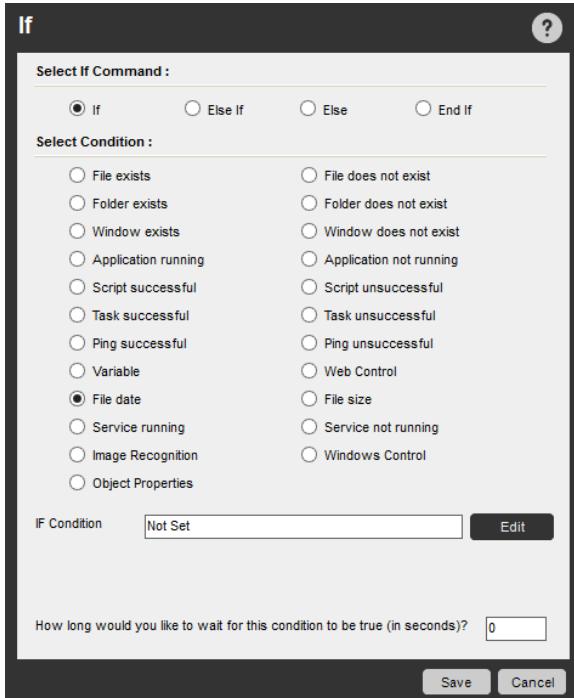
File Size Command

Use this command to check whether a file is larger, smaller, or the same as a size you specify.

File Date Command

Use this command to check the date and time that a file was created or last modified. You can specify a date range using 'between' or 'before' keywords. You can also specify a range using a number of days or a time schedule hh:mm:ss format.

Web Control Command

Use this command to identify whether a specific web control exists on a Web page. For example, your task can check for links, text boxes, and drop-down menus.

Use this command when editing tasks that are created using the Web Recorder.
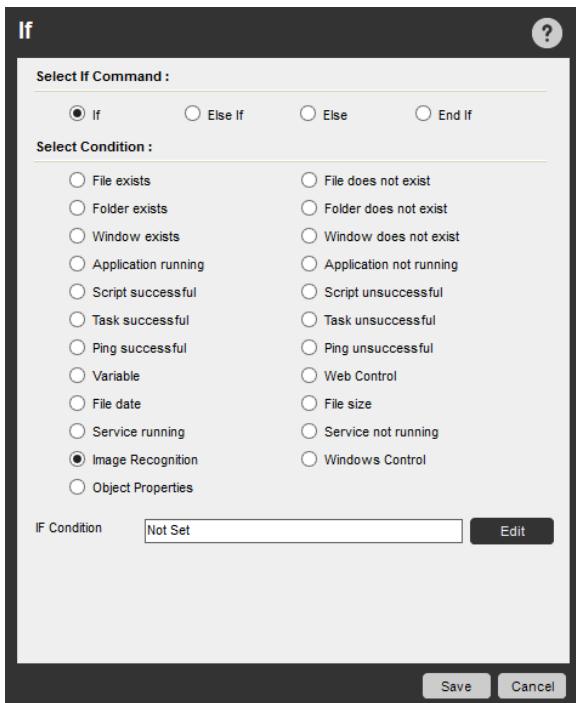
> Note: If you have upgraded to version 8.1.2 and change the parameter to 'Select Page', you will have to re-capture the control. Refer Editing a Web-only Task with Web Recorder Command for details.

Image Recognition Command

Use this command to check whether an image exists within another image. Automation Anywhere can recognize images that are stored in files or captured from a window when the task runs. You can even specify a minimum 'percentage match' for the If command to be true. Image Recognition also provides a powerful tool that enables you to click on an image if it is found within the larger image.
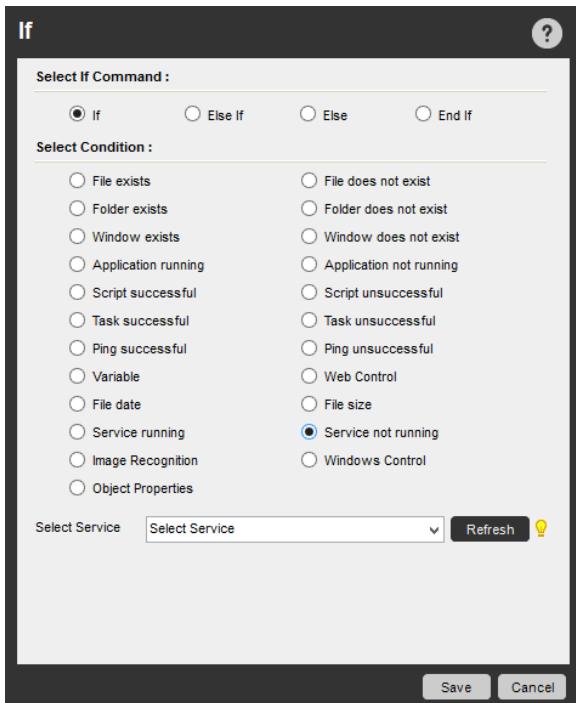
Service Running/Service Not Running Commands

Use these commands to check whether a particular service is running.
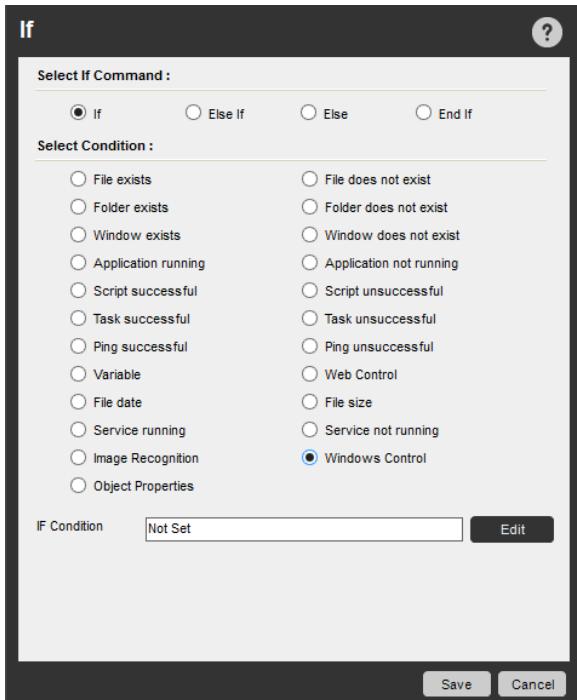
Windows Control Command

Use this command to check whether a specific control exists in an active window. For example, you might check for command buttons, list box items, text boxes, or menu tabs.

Setting a Wait Time for Conditions

Most of the conditional commands let you specify how long to wait for a condition to be true.

Let's say you want to perform an action only if a specific window exists. You know that it might take from 5 seconds to 5 minutes for the window to open. Obviously, you don't want to wait 5 minutes when the window comes up in just 30 seconds. Automation Anywhere takes care of this for you by following this model:

Using the IF command, you can select IF and 'Window does not exist,' and type the maximum time in seconds to wait for the window to be displayed. (In this case, 5 minutes, or 300 seconds.)

Automation Anywhere checks for the window to be displayed. If it appears in 30 seconds, Automation Anywhere immediately begins running the commands you specified within the conditional IF.

If you leave the 'How long would you like to wait...?' field empty, or enter '0 (zero)', Automation Anywhere checks for the condition immediately and runs the actions. If the condition does not exist, these actions are skipped and the actions that follow the ENDIF are run.

Also See...
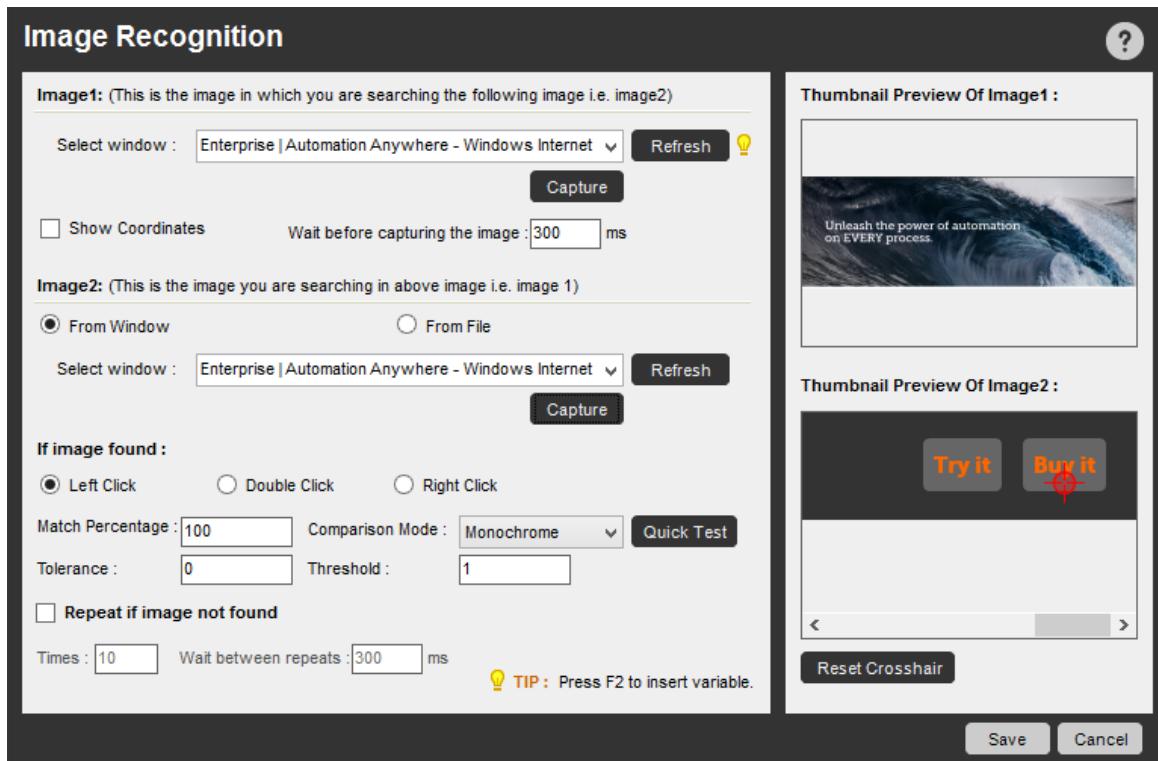
- If Endif Command - Examples
- Loop Command

Image Recognition Command

Feature also available in Standard edition version 8.1.2 or higher.

Image recognition command provides a visual approach to make your automation task work with the applications and interfaces with stunning accuracy. It acts as a "human eye" would act, accounting for the various parts of the image. Thus, Image recognition command makes it easy to divide and manipulate images.

Use this command to quickly search for a desired image within a source image and to readily automate any application that uses visual elements. Another major benefit of using image recognition is to locate an image after the image might be moved, for example, from one location on a web page to another.

Using the Image Recognition Command

To use the Image Recognition command, follow these steps:
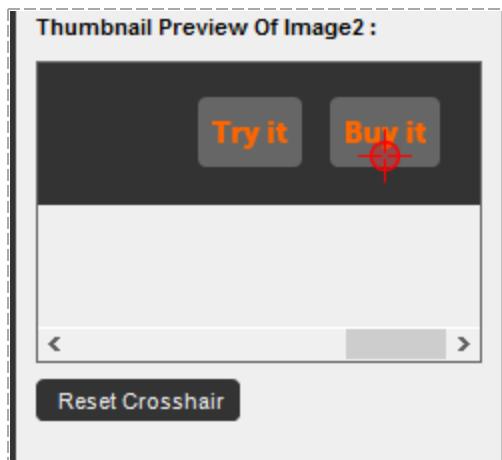
1. Select an image file from a folder or that is captured from an application window. This image can be standalone or contained within another image that is captured dynamically at run time.

2. To locate the image at run time, specify the minimum percentage match for which you want the command to return TRUE.

3. Select an option for clicking on the image when it is found within a larger image. You can select Left Click, Double-Click, or Right Click.

   If you are using the command for a window application, you also have the flexibility to position your click location. Just position the ' cross-hair* ' at the spot where you wish to click during run time when the image is found.

> Note: If using in conjunction with 'If – Else' command, select the option 'If the image is found, click on it'. for the crosshair to appear.

For larger images, you can scroll and position the 'cross-hair' as required.

Thumbnail Preview Of Image2 :

Try it   Buy it

Reset Crosshair

* Available from version 7.5.1 onward

> 💡 Tip: To get back to the default cross-hair position (center of the image), simply click
> 'Reset Crosshair'.

Select the options - Match Percentage, Tolerance and Comparison Mode:

- **Match Percentage:** A 0 (zero) Match Percentage means none of the pixels should mismatch between the two images. A Match Percentage of 20 means that even if up to 20% of pixels mismatch between the two images, the images will be considered to match. Use for images that are quite similar, yet are not matched.

- **Tolerance:** A 0% Tolerance means the individual RGB values of the pixels being compared must match, exactly. Similarly, a 20% tolerance could allow the individual RGB values of the pixels to vary from + 20% to - 20% for a successful match.

  a. While Match Percentage defines the extent of overall mismatch allowed between Image1 and Image2, Tolerance defines the extent of mismatch allowed between any two pixels under comparison.

  b. The RGB color space uses 8 bits each for red, green and blue and can have integer values ranging from 0 to 255.

  For instance when using Tolerance,

  - **Case1 - mismatch with 20% Tolerance -**

    Image1 pixel values: 250 240 238
    Image2 pixel values: 190 240 238

  - **Case2 - successful match with 20% Tolerance -**

    Image1 pixel values: 255 255 250
    Image2 pixel values: 220 230 200

    In Case1, the images are clearly a mismatch. The difference in R value of both images exceeds the Tolerance limit by 10 (20% of 255 is approximately 50 and 250 - 190 > 50). While in Case2 it's a successful match as the difference between the R,G and B values of both images fall within the Tolerance limit.

- **Comparison Mode:** Select one of the methods of comparison:

  a. **Normal mode:** Compares the RGB components of the pixels with the source. Use this when you are not bothered about the speed and want exact color match in complex scenarios.

  b. **Gray-scaled mode:** Converts pixels to an appropriate level of gray-scale prior to comparing. Use this when colors are of limited benefit in comparison and you want your test to run faster.

  c. **Monochrome mode with Threshold:** Converts pixels to either black or white prior to comparing. The threshold value is used while converting the RGB or Gray-Scaled pixels to either black or white (a pixel value less than the threshold value is black; greater is white). Use this when you want to recognize multi-colored text against multi-colored backgrounds.

- **A 'Quick Test':** Use the 'Quick Test' button to quickly ascertain the output is as required. This eliminates the need to run the entire test.

Quick Test

a. A 'Quick Test', being similar to an actual test, includes the parameters provided for Match Percentage, Tolerance and Comparison Mode.

b. If variables are used in the task, 'Quick Test' will not be applicable.

c. For tasks created in earlier versions (prior to 7.5), 'Quick Test' will work only after you recapture the image as this feature is available from version 7.5 onward.

💡 Tips and Tricks:
- Click on Image1 to preview in the default picture viewer.
- For faster image recognition, if a particular color dominates the entire screen (say gray), make sure that the starting pixel of image 2 (the one on the upper left corner) doesn't include that color.
- For Faster image recognition, ensure Image2 to be recognized is as small as possible.
- Use delay wisely.

Also See...
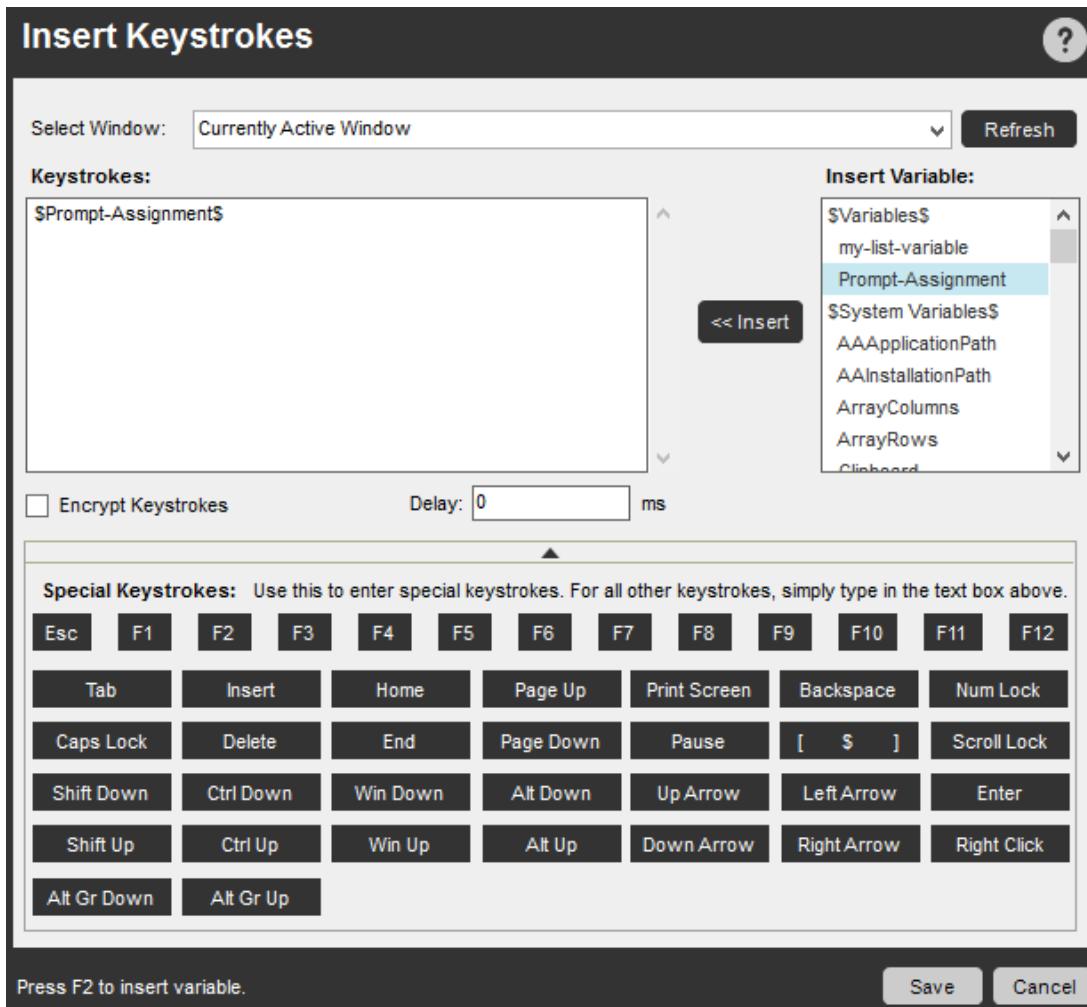
- Image Recognition Command - Examples
- Error Handling Command
- File and Folder Commands
- IF/Else Command
- Loop Command
- Variable Operation Command

Insert Keystrokes Command

Use this command to automate the typing of keystrokes in an application using your automation task.

To insert keystrokes, follow these steps:

1. Open the task in the Task Editor.

2. Double-click or drag the **Insert Keystrokes** command to the Task Actions List pane. The Insert Keystrokes window is displayed.

3. In the Select Window drop-down list, select the title of the application in which you want to type keystrokes.

> Note: You can insert a variable here by pressing the function key 'F2'.

Applications that you have already referenced in the task are listed at the top of the drop-down list. Applications that are currently open are listed below the dotted line in the list.
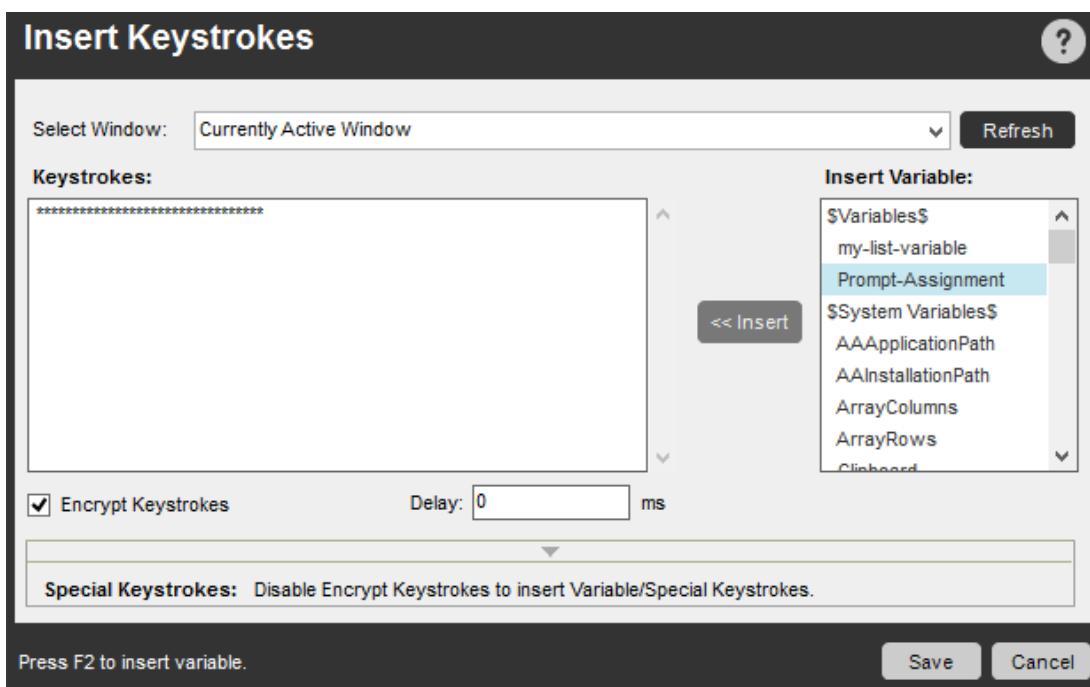
> Tip: Before using the Insert Keystrokes command, verify that the application window is open. If the application is not listed in the Select Window drop-down list, open the application and click the Refresh button in the Insert Keystrokes Select Window drop-down list.

4. In the Keystrokes text box, type (or cut/paste) the text you want inserted when the task runs.

   To insert special keystrokes such as Control, Shift, or Tab, click the button in the Special Keystrokes box.

   - For example, if you want the task to type Ctrl + C in an application, click Ctrl Down, type 'c,' then click Ctrl Up. The following commands and text in the Keystrokes window will be displayed: [CTRL DOWN]c[CTRL UP]. Similarly, if you want to type Ctrl + Shift + a, click Ctrl Down, Shift Down, type 'a,' and click Shift Up and Ctrl Up.

5. To prevent your tasks from being altered by unauthorized users, use the Keystroke Encryption option. This option hides sensitive information or passwords that are typed using the Insert Keystrokes command.

To prevent unauthorized users from turning off this option, password protect the automation task. For more information on security, see Setting Security Features for a Task.

6. If required, you can add a delay to this command.

> Note: The default delay is 0 (Zero) milliseconds.

You can even associate a variable with delay:

7. Click Save to save the command to your automation task.

Using Wild Cards

In the Select Window text box, you can type a wildcard character (*) at the start or end of the window title. This is helpful, for example, if you expect the window title to change over time. For example, you can type 'Microsoft Excel*' or '*Excel' or '*Excel*,' so that the window will still be selected when the task runs, even if the window title has changed to reflect the name of an open document. However, if you know the exact title of the window, we recommend that you type the complete window title to ensure that the actions will always performed only in that window.

Also See...
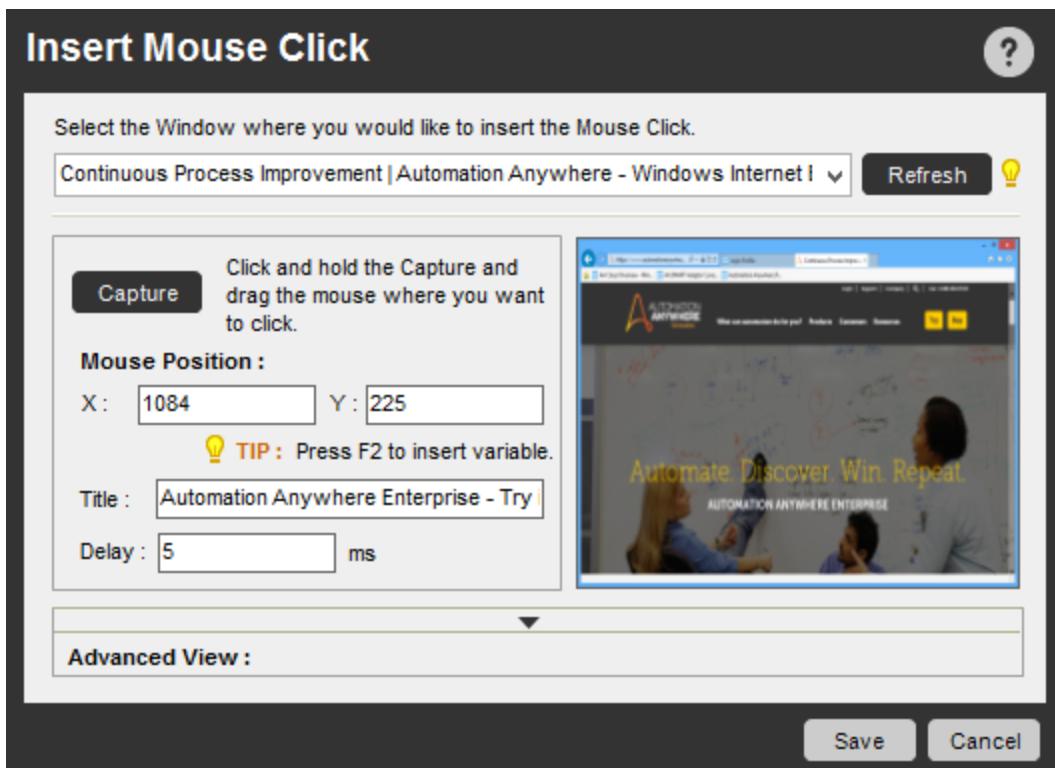
- Insert Keystrokes Command - Examples
- Insert Mouse Click Command
- Insert Mouse Move Command
- Insert Mouse Scroll Command

Other User Resources: User forums   Ask the Expert   Open a support case   Automation Examples

Insert Mouse Click Command

Use this command to automate mouse clicks in your automation task.

To insert a mouse click command, follow these steps:

1. Open the application in which you want to enter a mouse click.
2. In the Automation Anywhere Task List, select the task and click Edit.
3. In the Task Editor, drag or double-click the Insert Mouse Click command. The Insert Mouse Click window is displayed.

4. In the drop-down list at the top, all open applications are listed. Select the application in which you want to enter a mouse click. A small graphic image of the window you selected is displayed.

    If you do not see the application in the drop-down list, make sure the application is open and click Refresh.

5. **For Applications that change titles:** If the name of the application window title changes over time, you can use a wild card character (*) at the start or end of the window title that you specify. This ensures that the application is consistently found when running the task.

    For example, to ensure the Microsoft Excel application is always found, use 'Microsoft Excel*' or '*Excel' or '*Excel*'.

    Note: If you anticipate that the application title will never change, specify the exact name to ensure that the task references the correct application window.

6. Click Capture. Automation Anywhere activates the window you selected in step 4. Click in the window in the location where you want to insert the mouse click. Optionally, you can click Advanced View and set the following options:

    - Select the mouse button to click: Left Button, Middle Button, or Right Button.
    - Select the type of click: Click, Double Click, Button Down, or Button Up.
    - Select Relative To Screen or Window. Use the Window option when you work with varying screen resolutions (for example, a dual-monitor).

7. Click Save.

Other User Resources: User forums   Ask the Expert   Open a support case   Automation Examples

Also See...

- Insert Mouse Click Command - Examples
- Insert Keystrokes Command
- Insert Mouse Move Command
- Insert Mouse Scroll Command
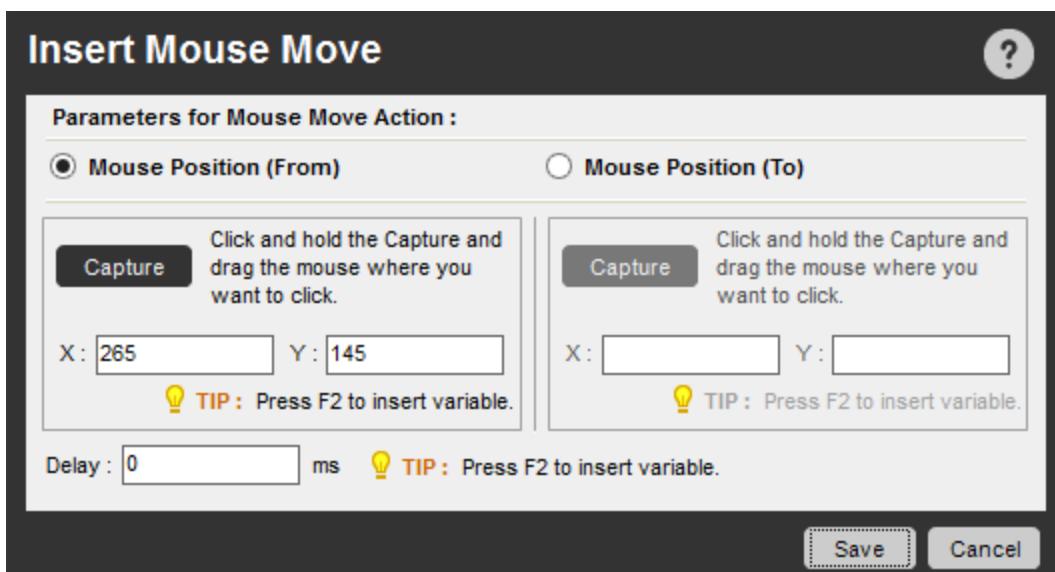
Insert Mouse Move Command

Use the Insert Mouse Move command to automate the movement of a mouse from one location to another.

The Mouse Move command is useful in automating menu navigation, such as when using the Windows Start menu, where moving the mouse over an item displays a new sub-menu. Inserting mouse clicks do not require moving the mouse to the click location.

To record mouse cursor movements, follow these steps:

1. Open the application in which you want to insert a mouse movement.
2. Open the task in the Task Editor.
3. In the Task Actions List, select the action after which you want to insert a mouse movement.
4. In the Commands list, double-click or drag the Insert Mouse Move command. The Insert Mouse Move window is displayed.
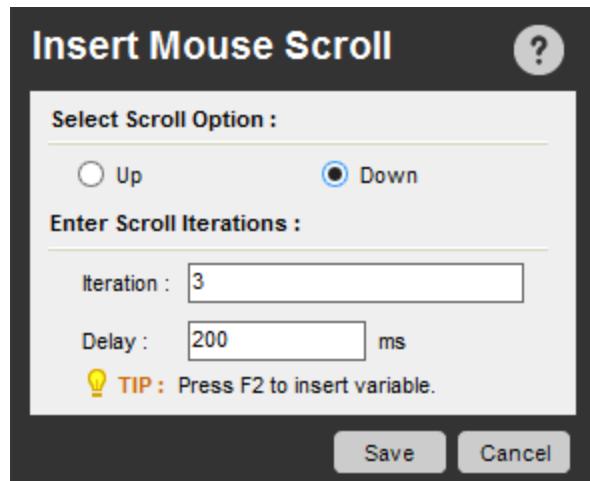
5. Select the Mouse Position (From) and click the Capture button to insert the mouse movement. Alternately, you can specify numeric values, or press the F2 key to insert variables for the X and Y coordinates in Mouse Position (From).
6. After you click on Capture, Automation Anywhere switches to the active window in the application.
7. In the active window, click on the left mouse button and drag the cursor to the location in which you want the mouse movement to begin. Release the mouse button. This step automatically captures the required information regarding the start location of the mouse movement.
8. In the Insert Mouse Move window, select the Mouse Position (To) and click on Capture. Alternately, you can specify numeric values, or press the F2 key to insert existing variables for the X and Y coordinates in Mouse Position (To). The yellow prompt bar will change to red if the content you click is not a table.
9. After you click on Capture, Automation Anywhere switches to the active window in the application.
10. Click on the left mouse button and drag the mouse cursor to the location in which you want the mouse movement to end. Release the mouse button.
11. Optionally add a delay in milliseconds.
12. Click Save.

Also See...

- Insert Mouse Move Command - Examples
- Insert Keystrokes Command
- Insert Mouse Click Command
- Insert Mouse Scroll Command

Insert Mouse Scroll Command

Use this command to automate the action of mouse scrolling. Use this command in conjunction with the Insert Mouse Click command.

To insert a mouse scroll command, follow these steps:

1. Select a the application window in which you want to insert a mouse scroll action.

2. Open the task in the Task Editor.

3. Double-click or drag the Insert Mouse Scroll command to the location required in the Task Actions List. The Insert Mouse Scroll window is displayed.

4. Select a scrolling direction option, either Up or Down .

5. Enter number of iterations.

6. Specify the delay time required.

7. Click Save. The mouse scrolling action is inserted in the task.

Also See...

- Insert Mouse Scroll Command - Examples
- Insert Keystrokes Command
- Insert Mouse Click Command
- Insert Mouse Move Command

Internet Connection Command
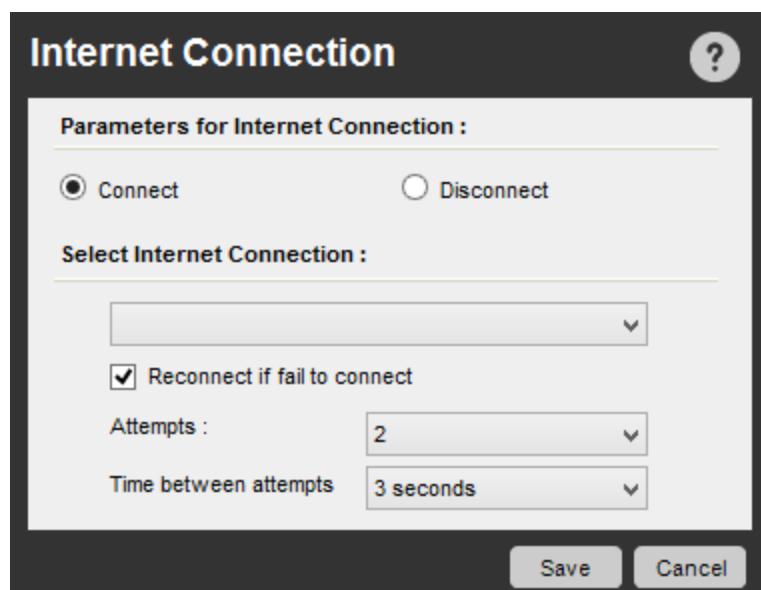
Use this command to have your task automatically connect to the Internet using a dial-up connection.

Using the Internet Connection Command

To insert an Internet Connection command in your task, follow these steps:

1. Double-click or drag the **Internet Connection** command to the Task Actions List pane. The Internet Connection window is displayed.

2. Select the dial-up connection from the drop-down list.

3. Specify whether you want to repeat the connection attempt if it fails. You can specify the number of times to attempt the connection, and the time interval between connection attempts.

4. Click Save to save the command to your automation task.

You can also insert a Disconnect command in the task to disconnect the dial-up connection.

Error Messages

If you do not select or specify a connection from the drop-down list, the program will display an error.

> Note: Only dial-up connections can be used with this command.

Also See...

- Internet Connection Command - Examples
- FTP-SFTP Command
- Launch Website Command
- Send Email Command
- Web Recorder Command
- Web Service Command
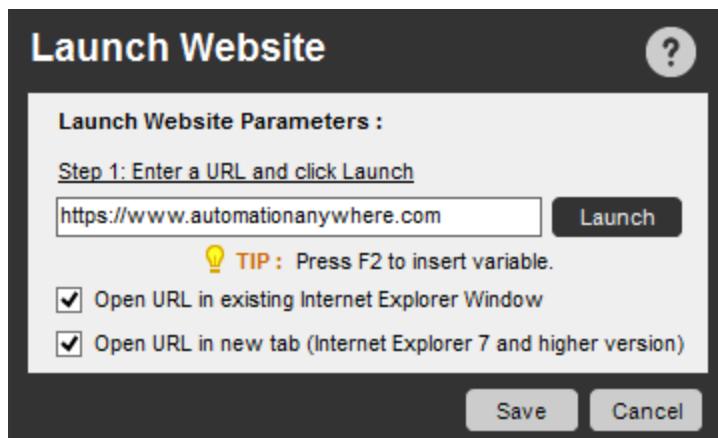
Launch Website Command

Use this command to launch a website from within an automated task.

Using the Launch Website Command

To launch a website from your task, follow these steps:

1. Open the task in the Task Editor.
2. Double-click on or drag the Launch Website command to the Task Actions List. The Launch Website window is displayed.
3. Specify the URL of the website to launch.
4. Click Launch.
5. Select the title of the website. If you do not see the title, click Refresh.
6. Save the command.

Launch Website ?

Launch Website Parameters :

Step 1: Enter a URL and click Launch

https://www.automationanywhere.com   Launch

💡 **TIP :** Press F2 to insert variable.

☑ Open URL in existing Internet Explorer Window

☑ Open URL in new tab (Internet Explorer 7 and higher version)

Save   Cancel

Also See...

- Launch Website Command - Examples
- FTP-SFTP Command
- Internet Connection Command
- Send Email Command
- Web Recorder Command
- Web Service Command
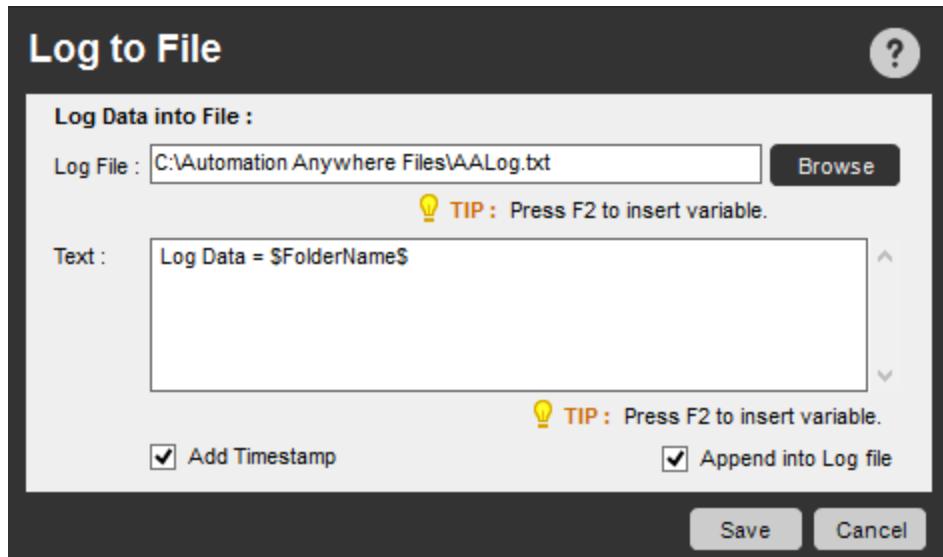
Log-to-File Command

Use the Log to File command to create a log file in which task information can be stored as the task runs.

Overview

Logging detailed task information is an important aspect of optimizing and debugging your automation tasks. Automation Anywhere provides an easy was for you to create a log file that is full of useful data about the events that occur as your task runs.

Log files can be saved in the several file formats, including .csv and .txt files. Encoding is provided for log files with options for ANSI, UNICODE, and UTF8.

The Log to File command helps you to do the following:

- Verify that a task has run properly, especially if the task has complex conditional and looping commands.
- Create a new log file if one does not already exist.
- Specify custom text to be included in the log file.
- Add a time stamp to the log file.
- Use a log file as a variable.

Also See...

- Log to File Command - Examples
- File and Folder Commands
- Windows Actions Command

Loop Command

Use this command to create loop structures in your automation task to automate repetitive tasks.

Overview

The Loop commands are some of the most powerful commands in Automation Anywhere. Four types of loop structures are provided:

- Start Loop
- End Loop
- Exit Loop
- Continue Loop

Sub-Commands

The LOOP command provides the following sub-commands:

- Start Loop
    - Times
    - List
    - Condition
    - Each Row in an Excel Dataset
    - Each Row in an Internet Explorer Table
    - Each Row in an SQL Query Dataset
    - Each File in a Folder
    - Each Folder in a Folder
    - Each Row in a CSV/Text File
    - Each Email Message on Server
    - Each Node in an XML Dataset
- End Loop
- Exit Loop
- Continue Loop

Using the Sub-Commands
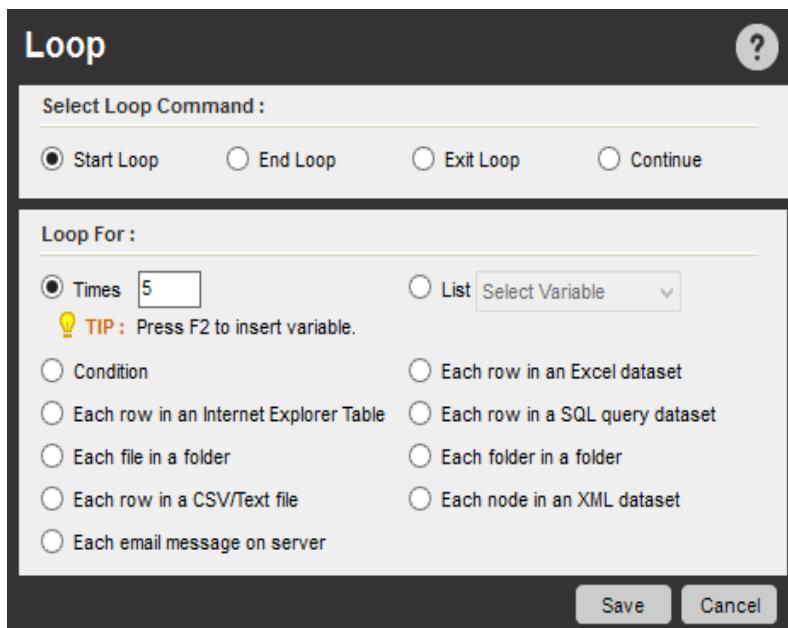
To insert an Loop command in your task, follow these steps:

1. Double-click or drag one of the **Loop** commands to the Task Actions List pane. The Loop window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.
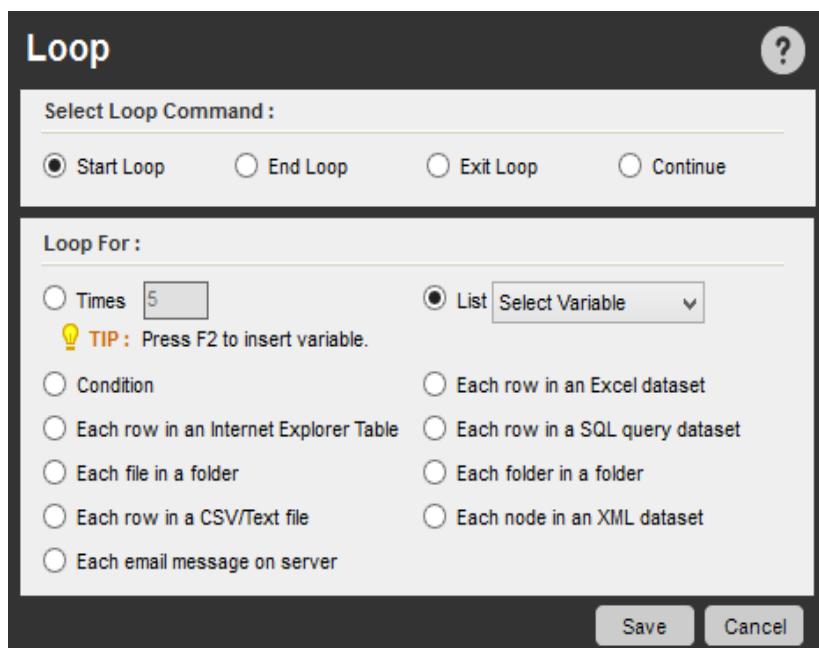
Start Loop Commands

The following commands are Start Loop commands:

1. **Times:** Use this command to loop for a number of times. The command loops the actions between the Start Loop and End Loop commands for the number of times you specify. For example, if you want to repeat the action 5 times, place the actions between Start Loop and End Loop commands and specify 5.

2. **List:** Use this command when you do not know how many times a task needs to loop. If you have a list variable that contains several values, you can use the List command to loop through the entire list until it reaches the end. Using this command, you can repeat the actions within the Start Loop/End Loop block as many times as necessary to reach the end of the list. You can access the list variable values within the loop. For example, the first time the loop runs you can get the first value of the list variable, the second time you get the second value of the list variable, and so on.

3. **Condition:** Use this command to run actions based on conditions that exist. For example, you might want to repeat a set of actions based on whether a file exists. You can check the condition at the end of the iteration by selecting the check box.

   Optionally, you can add a 'Wait' time for the condition to be true.

NEW! Using AND or OR operators in Loop While > Variable command*

* Available from version 7.5.2 onward

When you need to include the 'AND' or 'OR' conditions in conjunction with the 'Variable' sub-command, use the 'Add more conditions' option. It offers you the luxury of including all your 'AND' or 'OR' conditions within a single parameter, instead of specifying separate parameters for each condition.

Enable the 'Match Any' option for 'OR' conditions or 'Match All' for 'AND' conditions (refer image below)

> 📝 Note: You cannot include both; 'Match Any' + 'Match All'. You are required to specify them within a task, separately.

Similar conditions are used in If Variable command. Reference If-Else Command for details.

4. **Each Row in an Excel Dataset:** Use this command in conjunction with the **Get Multiple Cells** sub-command of the Excel com-mand. When you have retrieved a range of cells from Excel using `Excel > Get Cells > Get Multiple Cells`, the rows and columns of the range are stored in the system variable *$Excel Column$*.

For example, if you retrieve a range of cells from Excel that has five columns, you can access the five columns of the table by calling `$Excel Column(1)$, $Excel Column(2)$, $Excel Column(3)$, $Excel Column(4)$, $Excel Column(5)$`. This command is extremely useful for transferring data from one Excel spreadsheet to another, or to any other application.

5. **Each Row in an Internet Explorer Table:** Use this command in conjunction with the **Extract Table** sub-command of the Web Recorder command. When you extract a table by using the Web Recorder Extract Table command, the rows and columns of the table are stored in the system variable *$Table Column$*. For example, if you extract a table with five columns, you can access the five columns by calling `$Table Column(1)$, $Table Column(2)$, $Table Column(3)$, $Table Column(4)$, $Table Column(5)$`. This command is extremely useful for transferring data from a Web page table to other applic-ations, such as other Web pages or an Excel spreadsheet.

6. **Each Row in an SQL Query Dataset:** Use this command in conjunction with the Database commands. When you issue an SQL query using the Automation Anywhere Database SQL Query command, the result set for the SQL query is stored in the system variable *$Dataset Column$*.

For example, if you issue an SQL query to a database for five columns of data. You can access the five columns from the result set by calling `$Dataset Column(1)$, $Dataset Column(2)$, $Dataset Column(3)$, $Dataset Column(4)$, $Dataset Column(5)$`. This command is extremely useful for transferring data from a database to other applic-ations, such as a Web page or an Excel spreadsheet.

7. **Each File in a Folder:** Use this command to perform an action on all files in a folder. Often times, the number of files in a folder can change. You can specify a folder and loop through all files in that folder.

You can obtain information about the files and folder by using the following system variables:

- *$FileName$* - Use this variable to obtain the name of a file.
- *$Extension$* - Use this variable to obtain a file's extension.
- *$CurrentDirectory$* - Use this variable to obtain the path of the folder in which the filesa are stored.

All of these system variables return an empty value when used outside the looping command.

8. **Each Folder in a Folder:** Use this command to perform an action on one or more folders. To access the folder name within the loop, use the system variable *$FolderName$*.

9. **Each Row in a CSV/Text File:** Use this command to loop through each row in a CSV or text file.

10. **Loop For Each Message in Server:** Use this command to perform an action on each message that resides on your mail server. You might use this command to save mail attributes, such as Subject, Message, To:, Cc:, and Attachment.

11. **Each Node in an XML Dataset:** Use this command to loop through each node of an XML data set to obtain values from multiple nodes.

End Loop Command

Use this command to end the loop. The actions that exist between the Start Loop and End Loop commands will repeat based on the type of loop you have selected.

Whenever you insert a Start Loop command, an End Loop command is also inserted. If the End Loop is mistakenly deleted, you can insert an new one.

Exit Loop Command

Use this command to break out from a loop when a particular action or condition occurs. The Exit Loop command can exist only between the Start Loop and End Loop commands.

Continue Loop Command

Use this command to continue the next iteration of a loop when a particular action or condition occurs. When a Continue Loop command is reached, the commands up to End Loop are skipped and the next iteration of the loop is run.

Also See...

- If Else Command
- If Endif Command - Examples

Manage Windows Controls Command

Feature also available in Standard edition version 8.1.2 or higher.

Use this command to capture Windows control properties, such as buttons, list box items, text boxes, menu tabs, combo boxes, check boxes, and tables.

Using this command, you can perform actions on these controls or assign the captured properties to variables.

You can also use this command to simultaneously record actions that you perform.

The following list shows the Windows controls that you can capture and the actions that are available.

Control - Actions

1. Button - Click, Get Text

2. Edit (Text box) - Get Text, Set Text, Append Text, Click On Link

3. RadioButton - Get Radio Button Status, Select Radio Button

4. Checkbox - Set Check Box Status, Get Check Box Status

5. ComboBox/Drop Down - Get Total Items, Get Selected Index, Get Selected Item, Select Item by Index, Select Item by Text

6. Get Text - Click on link, Get Name of All Children, Get Value of All Children, Get Value Text, Get Name Text, Get Text

7. Tab / TabItem - Get Total Items, Get Selected Index, Get Selected Item, Select Tab By Index, Select Tab by Text

8. List / ListItem - Get Total Items, Get Selected Index, Get Selected Item, Select Item by Index, Select Item by Text

9. Menu/MenuBar/MenuItem - Get Total Items, Get Text by Index, Get Index by Text, Click Item By Index, Click Item by Text

10. Tree / Tree Item - Get Total Items, Get Selected Index, Get Selected Item, Select Item By Index, Select Item by Text

11. Table - Exports To CSV, Get Cell By Index, Get Cell By Text, Total Row, Total Column, Set Cell By Index, Set Cell By Text

The following actions are common to all Windows controls:

- Left mouse click
- Right mouse click
- Double-click
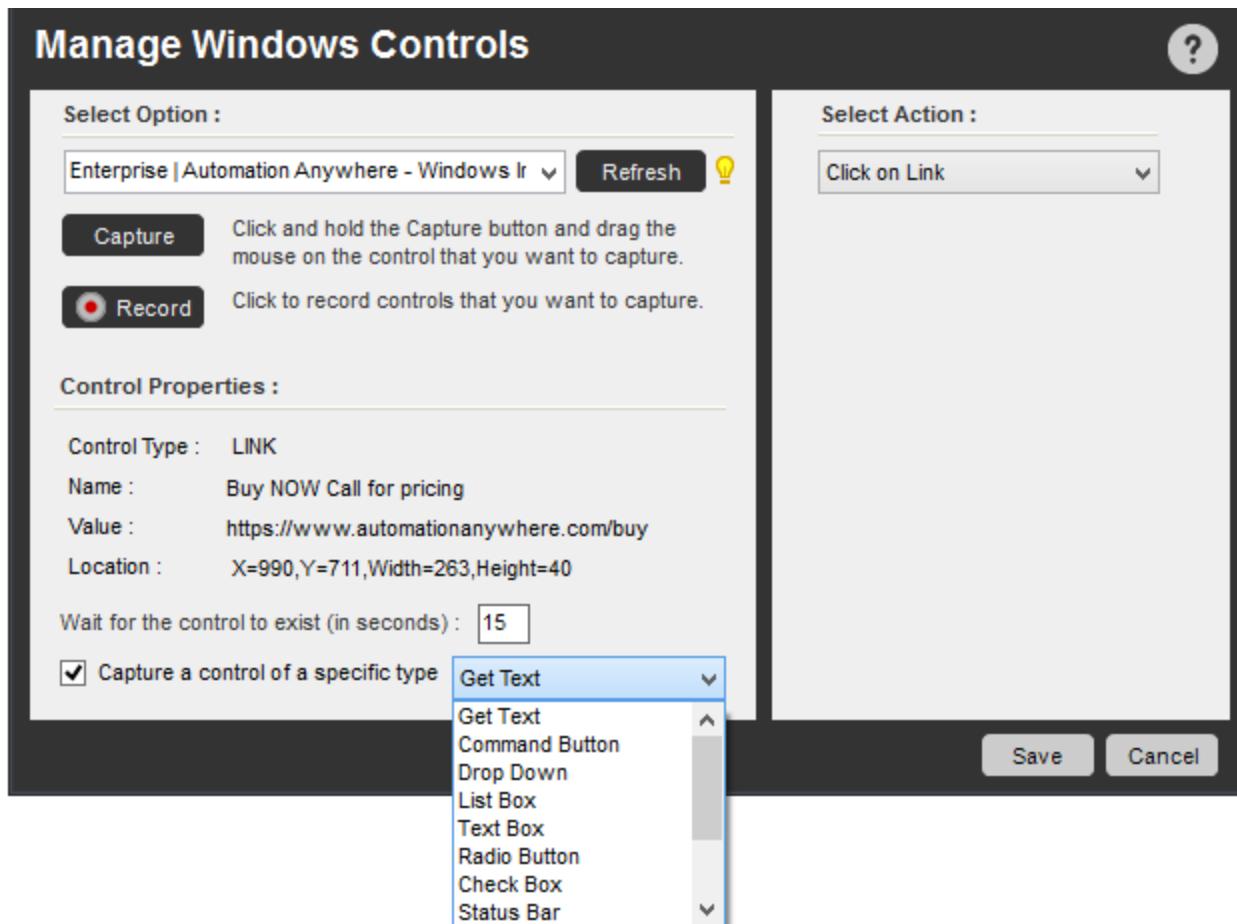
Capturing a Control of a Specific Type

Use the **Capture a control of a specific type** check box to select from a list of controls to capture. Only this control is captured when using this option. This option can help in avoiding errors that might occur when clicking on certain controls that possess different attributes.

The controls that are captured are displayed under **Control Properties**, showing the Control Type, Name, Value and its Location (coordinates).

> 📝 Note: Automation Anywhere can perform actions that use only standard Windows controls. If your application contains customized controls, Automation Anywhere might not be able to capture them.

Capturing Web Controls

To capture web controls, be sure to use the Manage Web Controls command, as opposed to the Manage Windows Controls command. Internet Explorer controls are recognized by the Manage Web Controls command.

Also See...

- Manage Windows Controls Command - Examples
- Log-to-File Command
- Windows Actions Command

Message Box Command
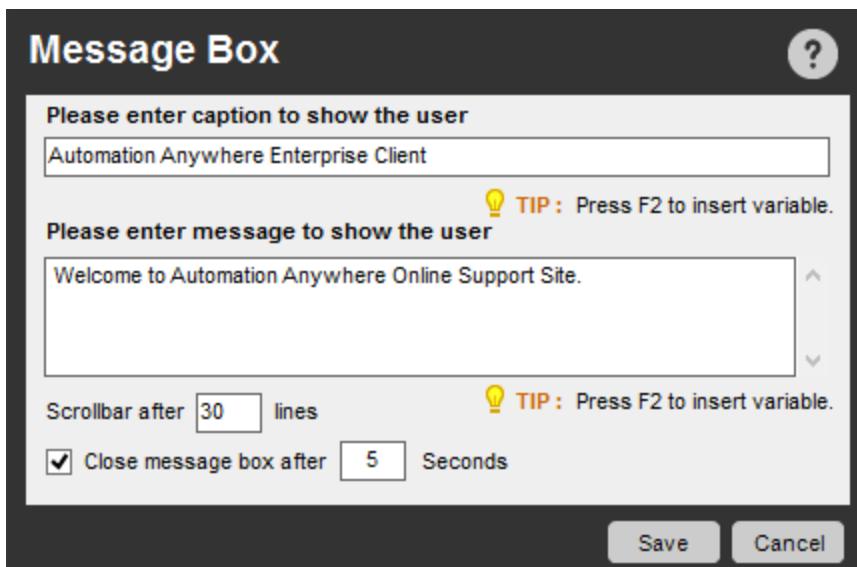
Use this command to insert a message box in an automated task.

Overview

The Message Box command enables you to insert a message box in a task when you want to display a message to the user when the task runs. You can specify a custom caption for the message box.

You can specify the following components in a message box:

- **Caption:** Enter a caption name for your message box.
- **Message:** Enter your message in the dialog box.
- **Scroll bar:** Insert a scroll bar in your message. The minimum message length for a scroll bar to be displayed is 10 lines.
- **Close Message Box:** Enter the seconds value after which the message box should close.

Common Use Case: Message boxes are useful for issuing a message to the user whenever the task concludes running. For example, when a web form task completes a run, you might issue a message that states: "Web Form Filled and Complete."

Also See...

- [Message Box Command - Examples](#)
- [Prompt Command](#)
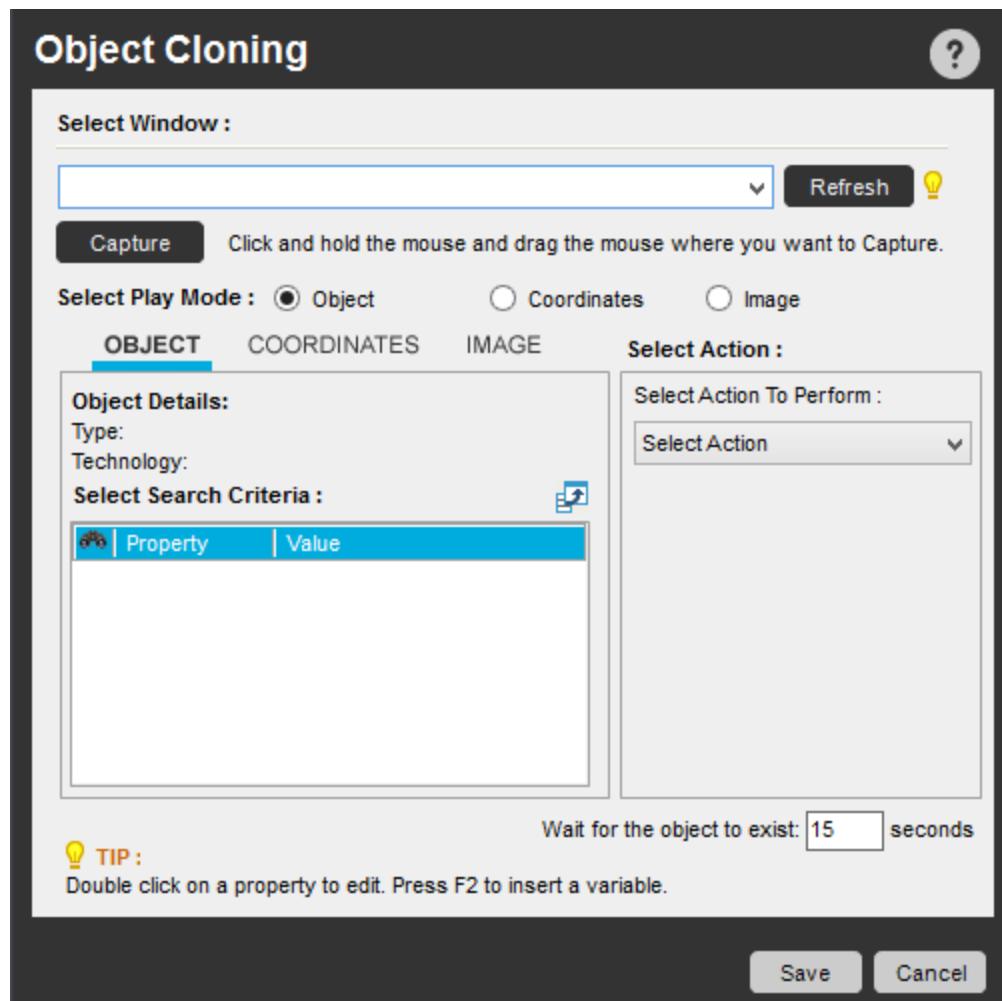
Object Cloning Command

Available with Enterprise edition only

In order that you can create and play tasks that use technology specific applications, Automation Anywhere offers the Object Cloning Command.

The Object Cloning Command is specifically useful if you wish to capture objects within windows that use technologies such as Flex, Silverlight, Java and Html etc.
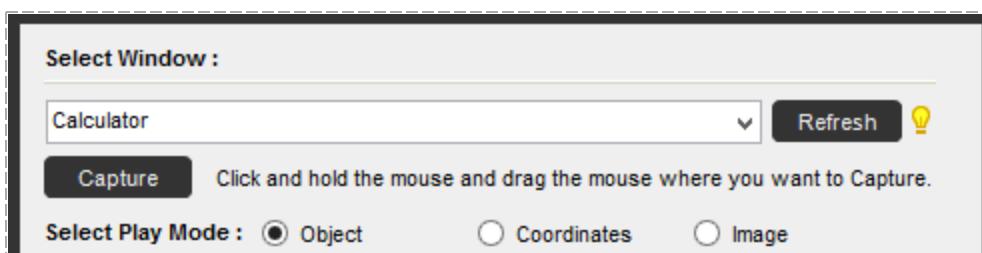
Using this command, you can capture objects, its co-ordinates and images. You can then select an action that can be performed on the captured object(s).

Selecting a Window

To capture an object and its parameters such as co-ordinates and image, you will be required to open the specific window and follow these steps:

1. Select the window from which you wish to capture the object(s).

💡 Tip: Click 'Refresh' if unavailable in the drop-down list.

2. Click 'Capture', hold the mouse and drag it to the object to be captured.

Capture

📝 Note: The selected object will be highlighted with a red border.

- The object is captured and displayed in the 'Object' tab.

OBJECT	COORDINATES	IMAGE

Object Details:
Type: MenuBar
Technology: Standard
Select Search Criteria :

	Property	Value
	Name	Application
	Type	MenuBar
	Path	3

📝 Note: The command captures the co-ordinates and image of the object, simultaneously.

Selecting the Play Mode

Coordinates: Use this option in 'Select Play Mode' if the objects are in fixed positions.

OBJECT	COORDINATES	IMAGE

Mouse Information :
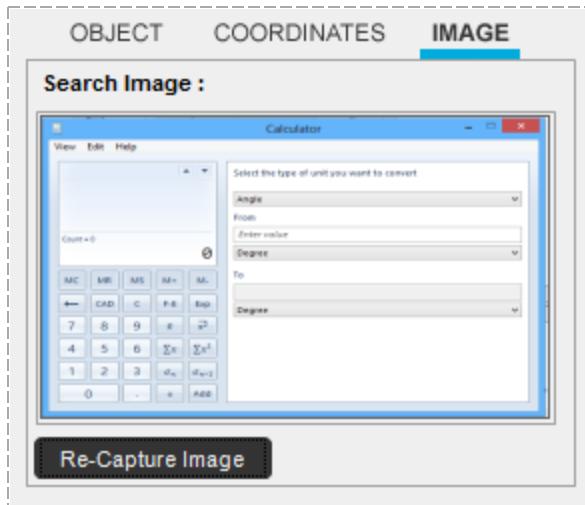
Position :

X : 305

Y : 49

Title : Application

Re-Capture Position

Image: Use this option, when the image is static.

Tip: Click 'Recapture Position/Recapture Image' in case of any discrepancy in captured co-ordinates or image.

 /

Selecting an Action to Perform

After capturing the object and its parameters, you can select an action that needs to be performed when running your task.

Note: Select Action to Perform will vary depending on the object or its coordinates captured.

Some of the options that are available to you are:

- Get Name - Usually applicable to text-box, button, label
- Get All Children Name
- Get Value
- Get All Children Value
- Click Actions - Left Click, Right Click, Double Click
- Append / Set / Get Text
- Get/Set Cell
- Set Cell by Index
- Set Cell by Text
- Get Cell Text by Index
- Get Cell Index by Text
- Get Total Columns
- Get Total Rows
- Export to CSV
- Get Property - Use this to capture object properties of multiple controls; namely Silverlight, Flex and Web applications. Assign the values of the captured object properties to a variable.

- Properties that can be captured using the Get Property action are:

Editing the Search Criteria

If you wish to change the properties of an object, you can do so by using the 'Search Criteria' option.

1. Click on 'Expand Search Criteria' icon - .

- The 'Object Cloning - Search Criteria' window pops up.

2. Change search criteria by adding or removing the search criteria icon - 👀 in the column prior to 'Property'.

> 📝 Note: You cannot set a search criterion for the property 'Path'.

Also See...

- Using Plugin Settings
- Using Java Settings
- If-Else Command
- Loop-While Command
- Object Cloning Command - Examples
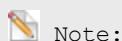
OCR Command

Available in Enterprise and Small Business editions only.

Overview

OCR (Optical Character Recognition) is a common technology for reading an image as a text file.

Use this command to capture an image or picture, convert it to text, and transfer the data to another application.

> **Note:**
> - The OCR command requires that Microsoft Office 2003 or later be installed on the computer.
> - For Microsoft Office 2007, ensure that the sub-component "Scanning, OCR, and Indexing Service Fil ter" (under Microsoft Office Document Imaging) is selected during installation.
> - If using Microsoft Office 2010 and above, you will be required to install the component separately; click here for details.

Sub-Commands

The OCR command provides the following sub-commands:

- **Capture Window -** Use this command to specify an application window title and capture the window as an image.
- **Capture Area -** Use this command to capture a specific area of an application window.
- **Capture Image by Path -** Use this command when text that you want to capture is contained in an image that is stored on your local computer or network drive. Specify a path name by typing it or browsing to the path.

  > Note: When running the automated task, the network drive that is specified must be accesible.

- **Capture Image by URL-** Use this command to capture an image located on a website URL.

Using the Sub-Commands

To insert an OCR command in your task, follow these steps:

1. Double-click or drag the required OCR command to the Task Actions List pane.
2. Select/input an appropriate window/area/path/url based on sub-command selected.
   Optionally, assign a variable by pressing F2.
3. Specify the OCR Engine; either TOCR or MODI.
   - By default MODI is selected. However, TOCR will be the default engine, if both installed.

> 💡 Tip: If you have upgraded to 8.0.1 and more than one OCR engine is installed to your system, it is recommended that you select the relevant OCR engine to ensure your existing tasks run fine.

4. Specify a threshold value for processing the image, either by dragging the bar or inputting the value in the text-box. This, to ensure that the captured text is as accurate as possible.

- For instance, if you wish to capture the text from the given image:
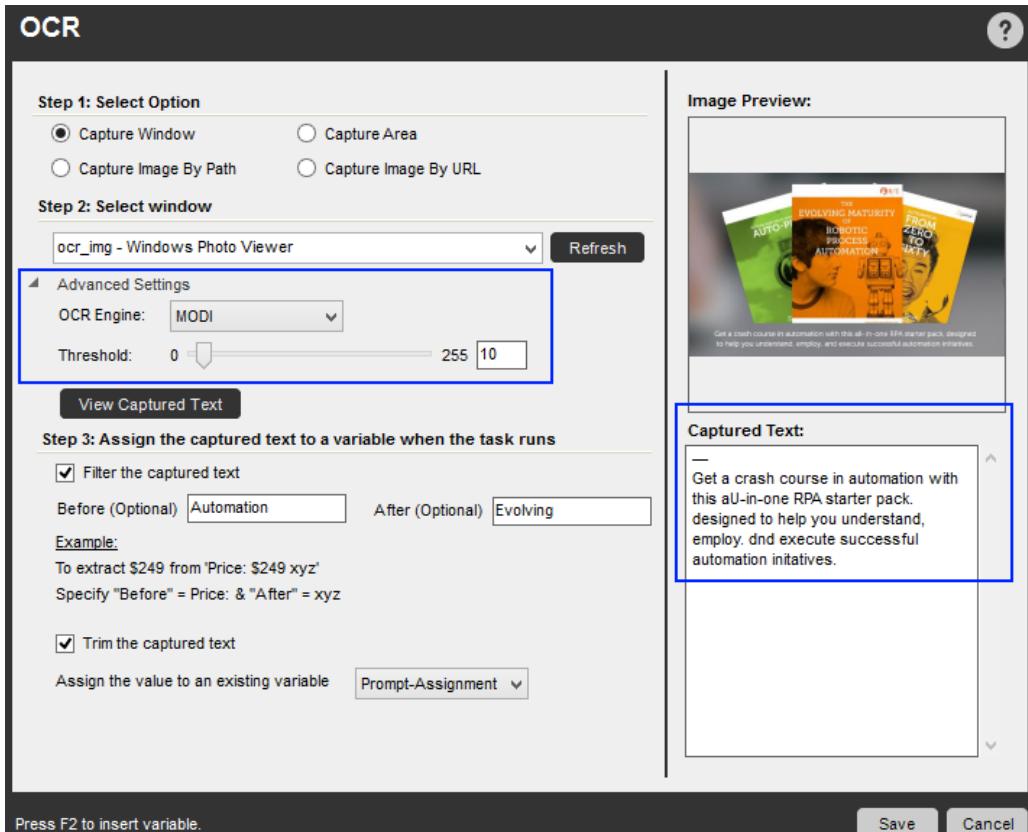
- In absence of the threshold option, the result could possibly be:

- However on specifying the threshold value, the result can be as accurate as possible:

5. View the Captured Text and adjust the threshold value till you achieve expected result.

6. Assign the captured text to a variable. Optionally, specify the parameters for filtering the text strings using 'Before' and 'After' and trimming leading/trailing spaces.

    - For example, in the text string `"Name: ABC Inc. Location:"`, if you require `"ABC Inc."`, specify Before = "Location" and After = "Name:".

7. Save the command to your automation task.

Using Wildcards (*)

You can specify wildcards (*) in a window title.

For example, use wildcards to perform an operation in any window that contains the word "Untitled" anywhere in the title. You can place the wildcard character before or after "Untitled" to capture variations, such as "Window 1: Untitled" or "Untitled - Text Document".

To capture all variations of "Untitled," simply specify `*Untitled*`.

Automation Anywhere searches for a window with a title of exactly "Untitled" first. If it does not exist, Automation Anywhere searches for windows with the word "Untitled" anywhere in the title.

Also See...
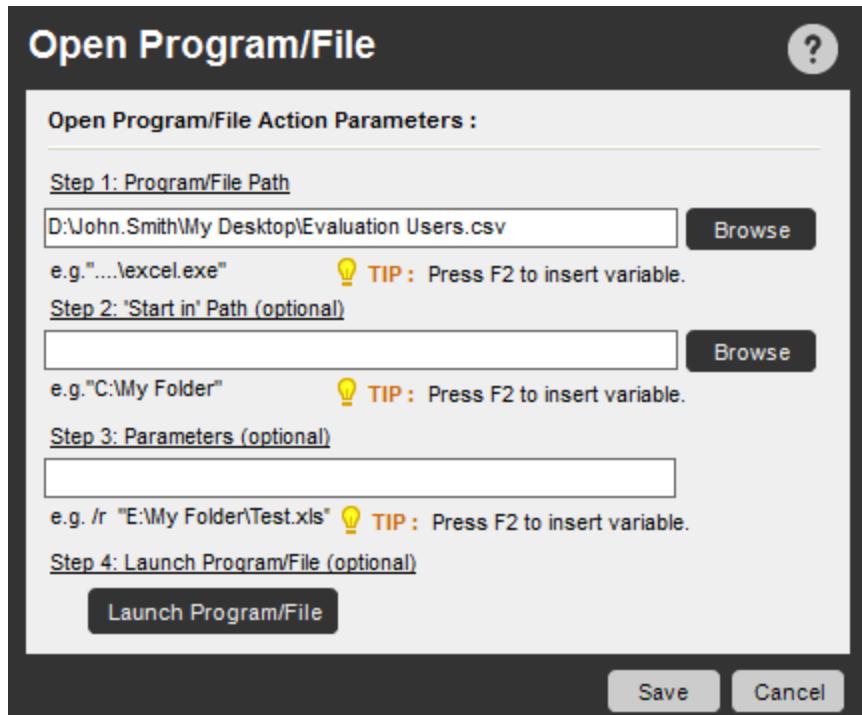
- OCR Command - Examples
- App Integration Command
- Database Command
- Email Automation Command
- Excel Command
- PDF Integration Command
- Read from CSV-Text Command
- Terminal Emulator Command
- XML Command

Open Program/File Command

Use this command to automatically launch a program or open a file.

You can perform the following actions in a task by using the Open Program/File command:

- If your program requires command line parameters, you can provide these parameters in the Parameters text box.
- If required, provide the directory path for your program in the 'Start in' Path text box. Automation Anywhere will verify that this path is valid.
- You can insert the current window title in a task.
- You can enter variables in a file path.

Open Program/File

Open Program/File Action Parameters :

Step 1: Program/File Path

| D:\John.Smith\My Desktop\Evaluation Users.csv | Browse |

e.g."....\excel.exe"   💡 **TIP :** Press F2 to insert variable.

Step 2: 'Start in' Path (optional)

| | Browse |

e.g."C:\My Folder"   💡 **TIP :** Press F2 to insert variable.

Step 3: Parameters (optional)

| |

e.g. /r "E:\My Folder\Test.xls" 💡 **TIP :** Press F2 to insert variable.

Step 4: Launch Program/File (optional)

Launch Program/File

Save   Cancel

Also See...

- Open Program/File Command - Examples
- File and Folder Commands
- Log-to-File Command
- Manage Windows Controls Command
- System Command
- Windows Actions Command

Pause Command

Use this command to pause an automated task at a particular point when it runs.

Overview

Inserting pauses into your tasks enables you to enter or modify information before a task completes a run.
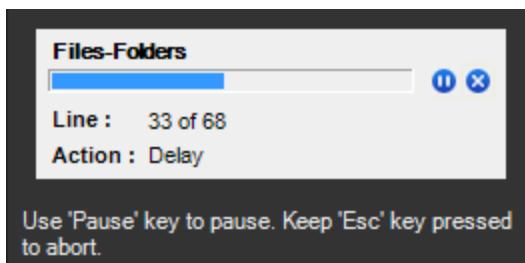
Insert a pause in your task when you need to check the status of attributes while the task is running. This is a great way to test and debug your tasks.

When a task reaches a pause, a Resume button is displayed. When you click on the Resume button, the task continues to run again.

Using the Pause Command

To insert a Pause command, follow these steps:

1. Open the task in the Task Editor.
2. In the commands list, under Task, drag a Pause command to the Task Actions List, in the location where you want to the task to pause when running. Or, you can double-click the Pause command to insert it at the current location.
3. A Pause command is inserted in the task. You can copy this command to add multiple pauses, or move the pause to another position in the task simply by dragging it.

Also See...

- Delay/Wait Command
- Run Task Command
- Stop Task Command

PDF Integration Command

Available in Enterprise edition only.

Overview

The Automation Anywhere PDF Integration command takes process automation to a new level. PDF files are pervasive on the web, in email, and across all sizes and shapes of business. It's become critical to automate routine tasks that involve manipulating and uploading these documents to databases, websites, and other repositories.

All sub-commands provide an option for overwriting existing output files. If you check the Overwrite check box, and the file name exists, the file is overwritten.

> 💡 Tip: It is recommended that you use variables with the command as many fields support variables.

Sub-Commands

The PDF Integration command provides the following sub-commands:

- PDF to Image
- Extract Form Fields
- Extract Text
- Merge Documents
- Split Document
- Encrypt Document
- Decrypt Document

Using the Sub-Commands

To insert a PDF Integration command in your task, follow these steps:

1. Double-click or drag one of the **PDF Integration** commands to the Task Actions List pane. The PDF Integration window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.

PDF to Image Command

Use this command to create separate images for each page of a PDF file. Follow these steps:

1. Select the PDF file using the Browse button. For encrypted PDFs, provide the user or owner passwords.
2. Specify a page range for the pages that you want converted to images.
3. Complete the Output Image Details section:
   a. **Type:** Select an output type from the drop-down list. The following types are available: TIFF, BMP, JPG, GIF, PNG, WMF, EMF, EXIF. For JPG output, specify a JPEG Quality.
   b. **Folder Path:** Specify a folder in which the images will be saved.
   c. **Image Prefix:** Specify an image name as a prefix for the image files. The files will be suffixed with index numbers.
4. Using the Advanced View section:
   a. Modify the X Resolution, Y Resolution, and Color for the output images. The default resolutions are 200 dpi.
   b. Select an output image type (either color or grayscale).
   c. Select a color property. The default is True Color (32 bits).
   d. For TIFF files, select a TIFF compression type.

PDF Integration

Select PDF Integration Command :

- ● PDF to Image
- ○ Extract Form Fields
- ○ Extract Text
- ○ Merge Documents
- ○ Split Document
- ○ Encrypt Document
- ○ Decrypt Document

PDF to Image

PDF Name : `D:\AAES\PDFInt\fw4_Filled.pdf`   [...]

	User Password :	Owner Password :
Password(s) : (If Protected)		

Page Range : ● All Pages   ○ Page(s) [   ]   (e.g. 1,3,5-12)

Output Image Details :

Type : `JPEG`   JPEG Quality : `100`

Folder Path : `D:\PDF_Images`   [...] 💡

Image Prefix : `FW4`

(Output Files will be created as prefix_1.type,...)

☐ Overwrite File(s)

💡 **TIP :** Press F2 to insert variable.

X Resolution (in dpi) : `300`

Y Resolution (in dpi) : `300`

Type : ● Color   ○ Grayscale

Properties : `True Color (32 bits)`

TIFF Compression Types :
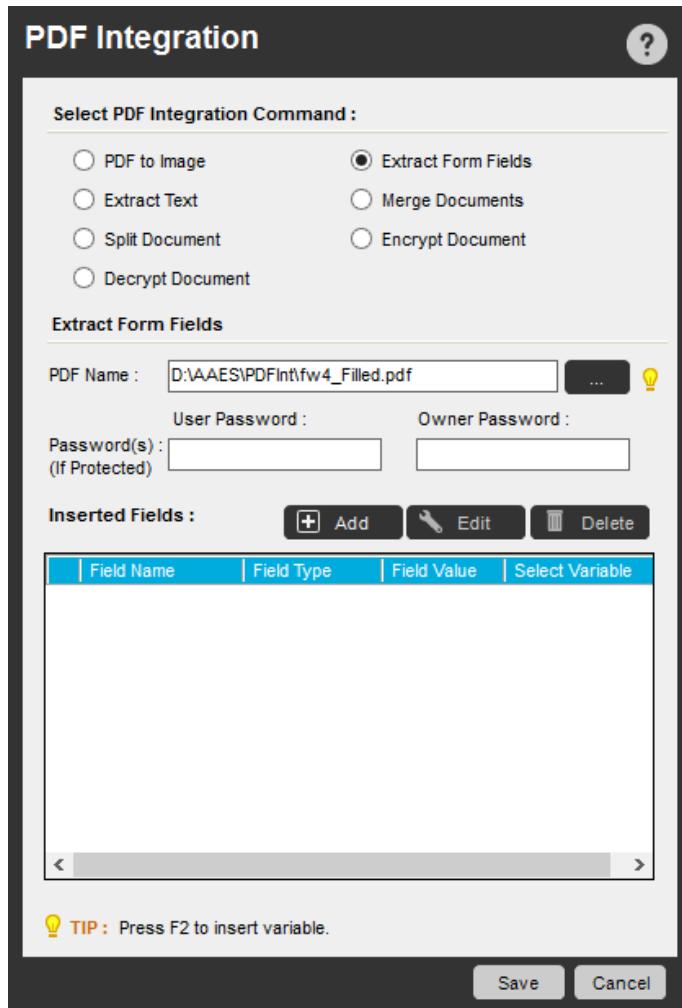
- ● None
- ○ LZW
- ○ RLE
- ○ CCITT Group 3
- ○ CCITT Group 4

← Normal View

Save   Cancel

Extract Form Fields Command
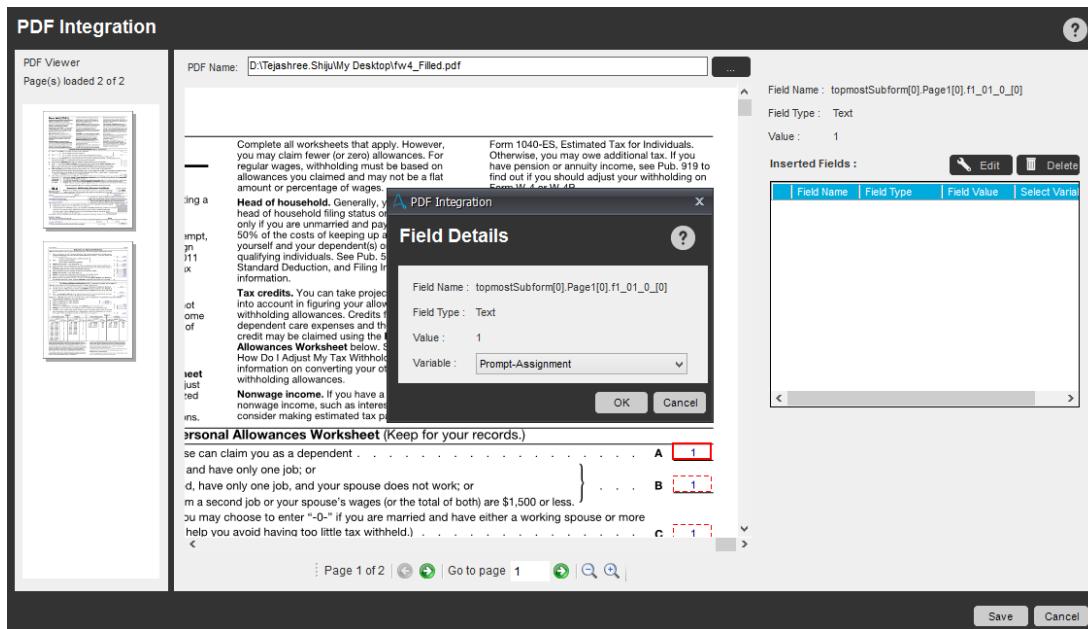
Use this command to extract all form field data and selected text to variables. Follow these steps:

1. Select the PDF file and click the Add button. A PDF Viewer window is displayed, showing all of the fields as dotted red boxes.

PDF Integration

Select PDF Integration Command :

- ○ PDF to Image
- ○ Extract Text
- ○ Split Document
- ○ Decrypt Document
- ● Extract Form Fields
- ○ Merge Documents
- ○ Encrypt Document

Extract Form Fields

PDF Name : `D:\AAES\PDFInt\fw4_Filled.pdf` [...]

Password(s) :
(If Protected)
- User Password :
- Owner Password :

Inserted Fields : [+ Add] [✎ Edit] [🗑 Delete]

Field Name	Field Type	Field Value	Select Variable

💡 **TIP :** Press F2 to insert variable.

[Save] [Cancel]

2. Right-click and select Add Field - here a solid red box is visible.
3. The Field Form is displayed, where a variable is assigned to the field. Click OK to view the inserted fields on the right side table of Inserted Fields.

4. You can also select particular text to store in a variable, if required.
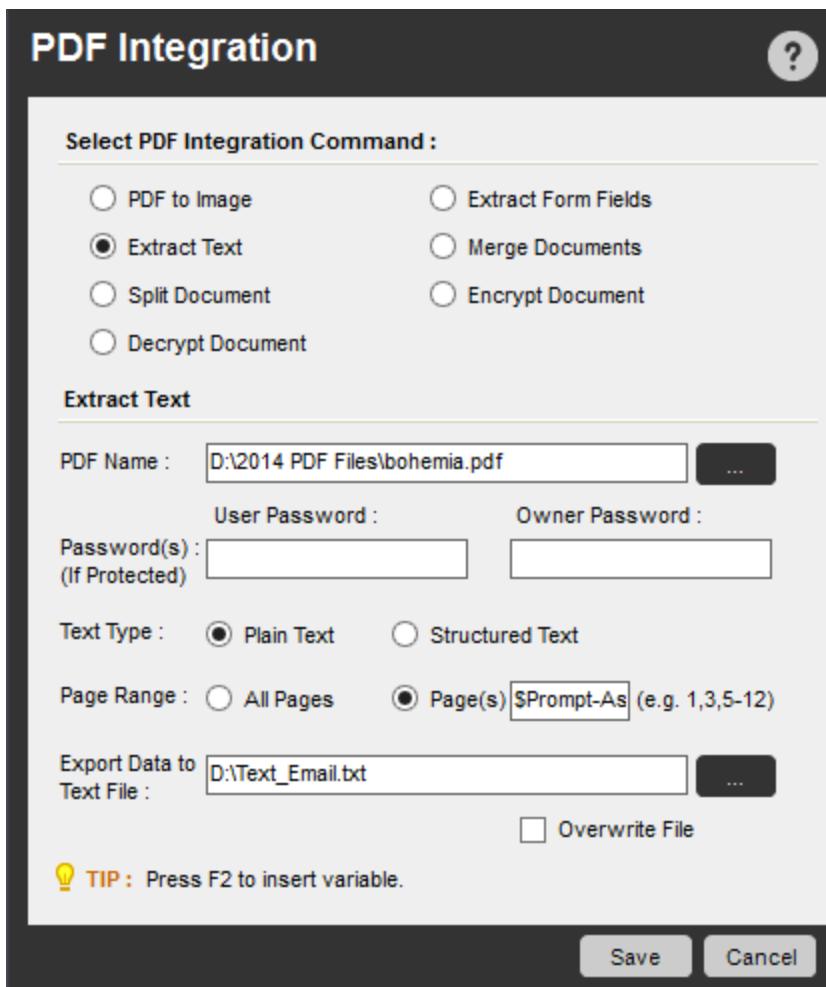
5. Click Save to view the inserted fields in the command. Modify them by using the Add, Edit, and Delete buttons.

6. Save the command. The inserted field values are stored in the variables when the task is run.

Extract Text Command

Use this command to extract text from a PDF to a text file. Follow these steps:

1. Select the PDF file using the Browse button. Enter passwords for encrypted PDF files.

2. Select a text type:
   - Plain Text: Similar to a simple copy/paste of the entire PDF file
   - Structured Text: Preserves the formatting from the original PDF file

3. Specify a page range.

4. For Export Data To Text File, specify an output file name with a .txt extension.
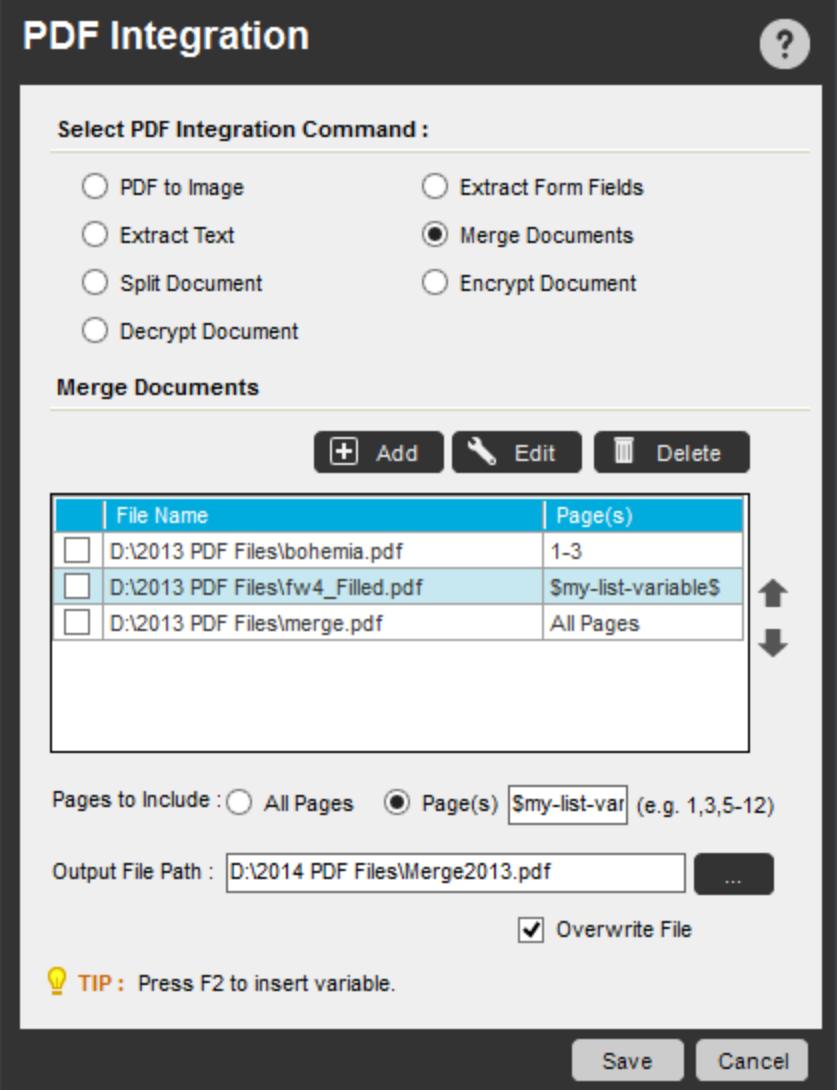
Merge Documents Command

Use this command to merge selected PDF documents into a single PDF file. Both encrypted and unencrypted PDFs are supported. Follow these steps:

1. To select the PDF files for merging, click the Add button. For encrypted PDFs, user or owner passwords must be specified.

2. For each PDF file added, specify the Pages to Include. After all files are added, you can modify the pages to include by selecting each PDF file.

3. Specify an output file path for the merged PDF file.

Split Document Command

Use this command to split a PDF file. Follow these steps:

1. Select the PDF file to be split and enter user or owner passwords if the file is encrypted.

2. Specify the Output File Creation options:
   - Pages Per File: Specify how many pages each new output file will contain (such as 5 pages for each split).
   - Single File With Page Range: Select this option to create a single output file that contains the page numbers specified.
   - Blank Page As A Separator: When selecting this option, a new output file is created whenever a blank page appears within the PDF document.
   - Bookmark Level Per File: Splits the file according to the bookmark level that the PDF contains. If 3 levels are present, the drop-down will show Level1, Level2, and Level3 as options. Split the PDF by selecting the appropriate bookmark level.

3. Specify Output File Details:
   - Folder Path: Specify a folder location in which all output files will be saved.
   - File Prefix: Specify the file name, with a .pdf extension, to be used for each output file (and suffixed using index numbers).
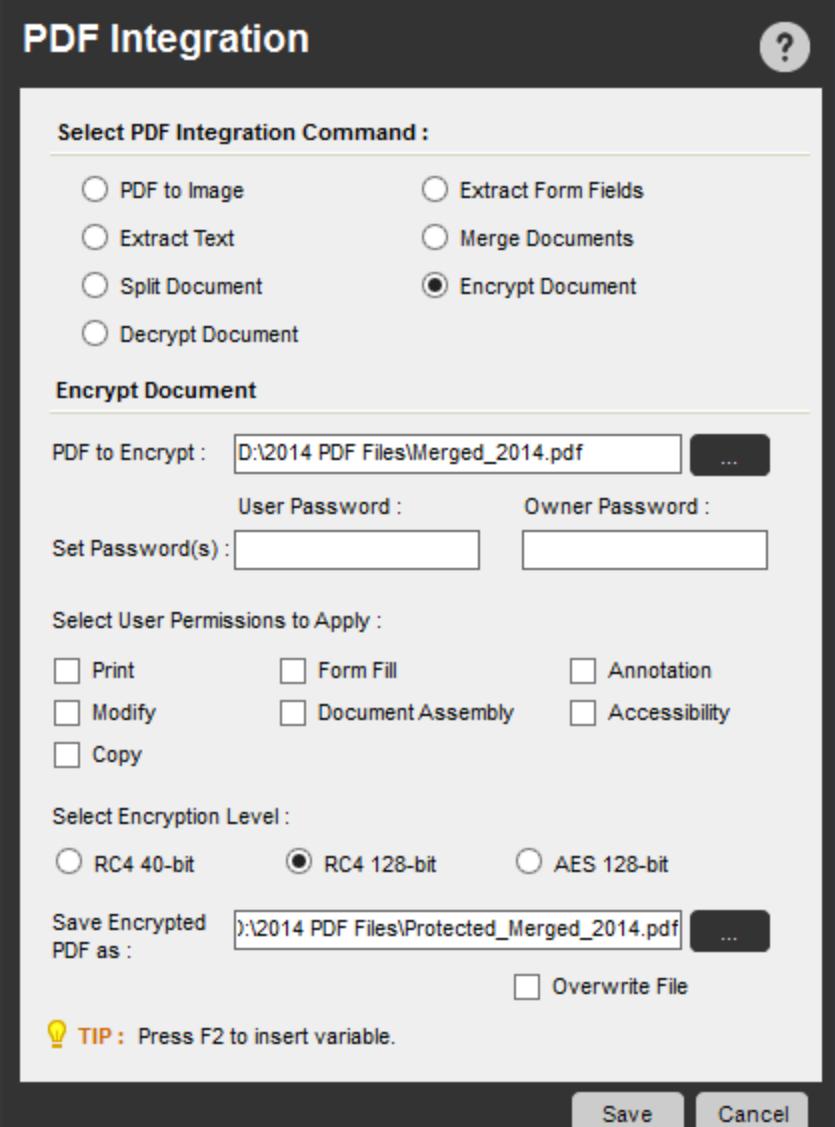
Encrypt Document Command

Use this command to encrypt PDF documents and grant permissions to users. Follow these steps:

1. Select a PDF you want to encrypt and set user and owner passwords for the encrypted file.

2. Select one or more permissions to apply to the encrypted document for the user.

3. Select an encryption level (RC4 40-bit, RC4 128-bit, or AES 128-bit).

4. Specify the name of the output file, with a .pdf extension.

PDF Integration

Select PDF Integration Command :

- ○ PDF to Image
- ○ Extract Text
- ○ Split Document
- ○ Decrypt Document
- ○ Extract Form Fields
- ○ Merge Documents
- ● Encrypt Document

Encrypt Document

PDF to Encrypt : `D:\2014 PDF Files\Merged_2014.pdf`   `...`

Set Password(s) :
- User Password : [   ]
- Owner Password : [   ]

Select User Permissions to Apply :

- ☐ Print
- ☐ Modify
- ☐ Copy
- ☐ Form Fill
- ☐ Document Assembly
- ☐ Annotation
- ☐ Accessibility

Select Encryption Level :

- ○ RC4 40-bit
- ● RC4 128-bit
- ○ AES 128-bit

Save Encrypted PDF as : `):\2014 PDF Files\Protected_Merged_2014.pdf`   `...`

☐ Overwrite File

💡 **TIP :** Press F2 to insert variable.

[Save] [Cancel]

Decrypt Document Command

Use this command to decrypt a PDF document that was previously encrypted. Follow these steps:

1. Select an encrypted PDF document to decrypt, and enter the user/owner password.
2. Specify an output file name with .pdf extension to save the decrypted PDF. The password security is removed for the decrypted file. All permissions that existed for the encrypted file remain, without password protection. This ensures that all document

restrictions remain for security in the newly created decrypted file.

Also See...

- PDF Integration Command - Examples
- App Integration Command
- Database Command
- Email Automation Command
- Excel Command
- Loop Command
- OCR Command
- Printer Settings Command
- Read from CSV-Text Command
- Terminal Emulator Command
- XML Command

PGP Command

Available in Enterprise edition only.

Use the PGP (Pretty Good Privacy) command to automatically encrypt and decrypt files for security.

Overview

The PGP command helps you to automate the process of encrypting and decrypting files.

The command supports both types of encryption: symmetric (passphrase) and asymmetric (public/private keys):

- **PGP Passphrase:** Requires a pass phrase to encrypt and decrypt.
- **PGP Public/Private Key:** Requires a PGP public and private key file to encrypt and decrypt.

Sub-Commands

- Encrypt Files
- Decrypt Files
- Create Keys

Using the Sub-Commands

To insert a PGP command in your task, follow these steps:

1. Double-click or drag one of the **PGP** commands to the Task Actions List pane. The PGP window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.

Encrypt Files Command

Use this command to encrypt one or more files using one of the encryption methods: either passphrase or public/private key.

To encrypt files, follow these steps:

1. Select the 'Passphrase' encryption type and enter a pass phrase. Alternatively, select the Public/Private Key encryption type. For this type, locate or specify the Public Key File path. Create public key if necessary.

   All other options are the same for both encryption types.

2. Select a symmetric algorithm. The algorithm options are:

   - Idea
   - TripleDES
   - CAST5
   - Blowfish
   - AES128
   - AES192
   - AES256
   - Twofish256

3. Locate or specify the source file path.

4. Locate or specify the destination file path.

5. Check the option for overwrite files/folder, if necessary.

6. If required, append the extension for the destination file. For example, .enc.

7. Compress the encrypted files using one of the compression algorithms:

   - Zip
   - Bzip2
   - Zlib

8. If required, check the Armor data (text output) option to enable ASCII Armor output.

> 🔎 Tip: For Source or Destination path, use '*.*' or '*.txt' to select multiple files in a folder.
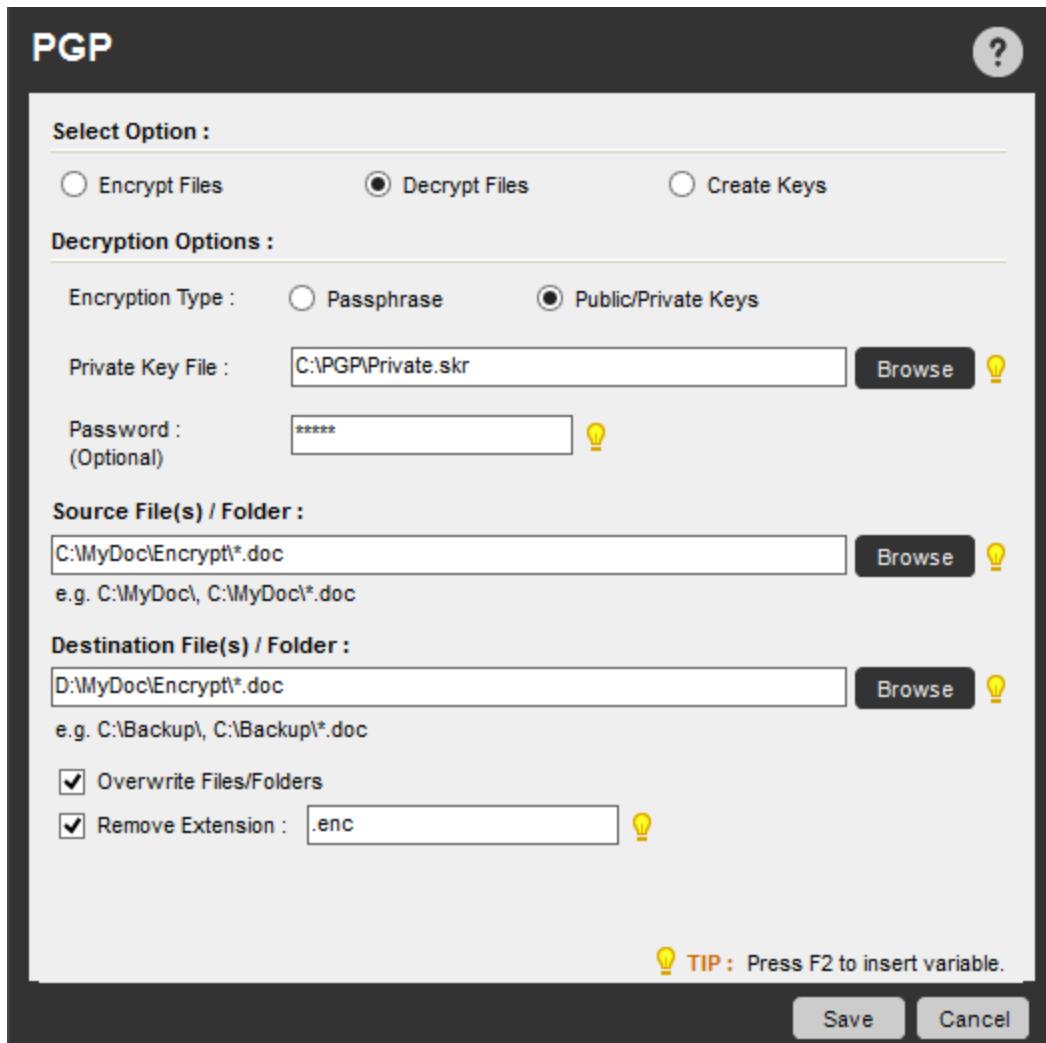
Decrypt Files Command

Use this command to decrypt one or more files that have been previously encrypted. You can decrypt files that are encrypted with either Passphrase or Public/Private Key encryption type.

To decrypt a file, follow these steps:

1. Select the Passphrase encryption type, which requires a specific pass phrase. Alternatively, select the Public/Private Key encryption type. For this type, locate or specify the Private Key file. While using password-protected private keys, enter the password.
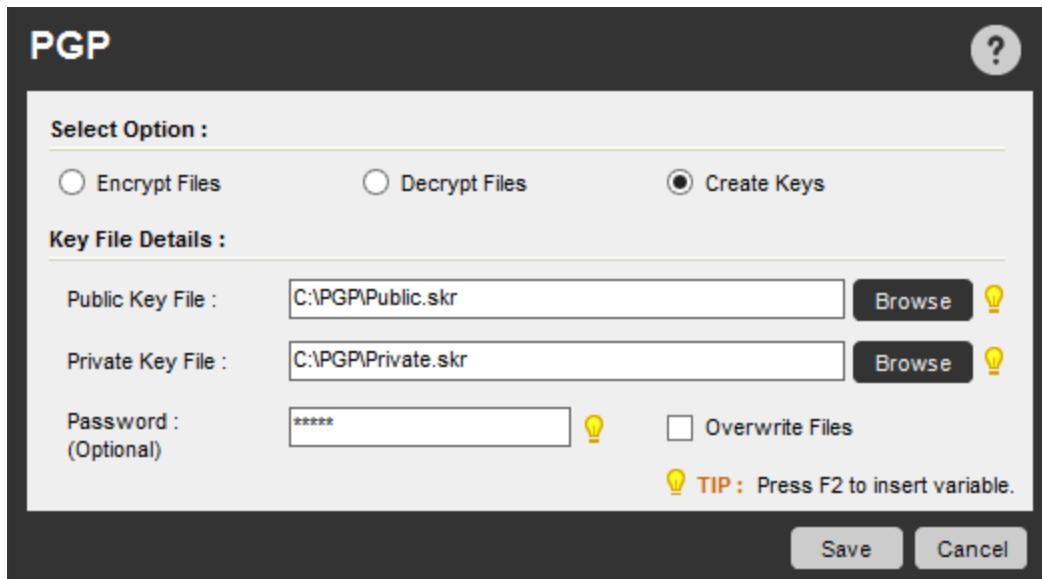
   All other options are same for both encryption types.

2. Enter the pass phrase for decryption.
3. Locate or specify a source file path for decryption.
4. Locate or specify a destination file path.
5. If required, check the option to overwrite files/folder.
6. If required, select the Remove extension option.

Create Keys Command

Use this command to create public and private encryption keys. You can add additional security to the keys by specifying a password that encrypts the keys.

To create new keys, follow these steps:

1. Select the Create Keys option.
2. Locate or specify a Public Key File.
3. Locate or specify a Private Key File.
4. Optionally specify a password that encrypts the key files.
5. If required, select the Overwrite Files option.

Also See...

- Setting Security Features for a Task
- PGP Command - Examples

Play Sound Command

Use this command to insert and play sounds in your automated tasks.

Sub-Commands

The Play Sound command provides the following sub-commands:

- Play Beep
- Play Media File

Overview

You can insert a Play Sound command in your automated task to play a beep or another media file before/after it runs a particular command.
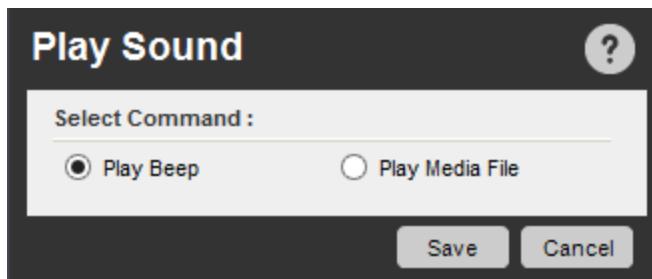
Using the Sub-Commands

To insert a Play Sound command in your task, follow these steps:

1. Double-click or drag one of the **Play Sound** commands to the Task Actions List pane. The Play Sound window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.
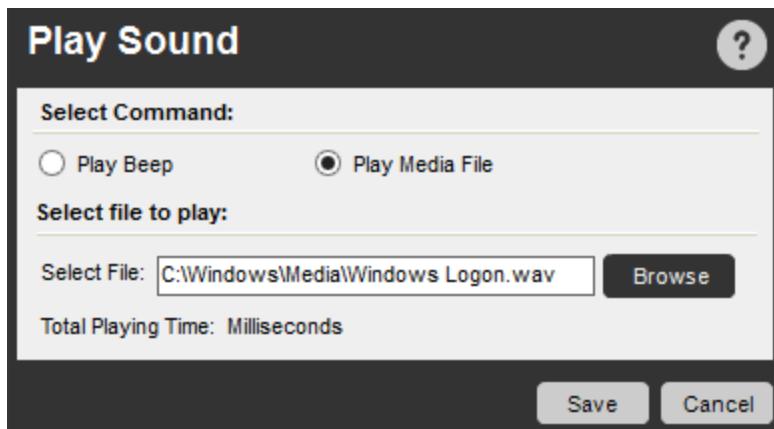
Play Beep Command

Use this command to play a beep sound when the task encounters the Play Beep command. You can use a beep sound to alert users when particular actions take place, or to alert you when prompted for input.

Play Media File Command

Use this command to play a media file in your automation task. When the task plays the media file, it does not proceed to the next automation task command until the media file finishes playing.

Also See...

- Play Sound Command - Examples
- Clipboard Command
- Comment Command
- Error Handling Command
- Image Recognition Command
- Prompt Command
- Screen Capture Command
- String Operation Command
- Variable Operation Command

Printer Settings Command

Use this command to automate printer settings in your task.

Overview

One of the most manually intensive activities of any IT staff involve managing printer settings for individuals and groups. Using the Printer Settings command, you can automate many of these tasks.

Sub-Commands

The Printer command provides the following sub-commands for managing printer settings:

- Get Default Printer
- Set Default Printer
- Remove Printer

Using the Sub-Commands

To insert a Printers command in your task, follow these steps:

1. Double-click or drag one of the **Printers** commands to the Task Actions List pane. The Printers window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.

Get Default Printer Command

To get the default printer using your automated task, follow these steps:

1. Open the Task Editor.
2. Double-click the Get Default Printer command.
3. Click Save.

Set Default Printer Command

To change the default printer using your automated task, follow these steps:

1. Open the Task Editor.
2. Double-click the Set Default Printer command.
3. Select a printer from the list of installed printers.
4. Click Save.

Remove Printer Command

To remove a printer using your automated task, follow these steps:

1. Open the Task Editor.
2. Double-click the Remove Printer command.
3. Select a printer from the list of installed printers.
4. Click the Run Script command.
5. Click Save.

Also See...

- Printer Settings Command - Examples
- Active Directory Command
- System Command
- SNMP Command

Prompt Command

Use this command to insert a user prompt that requests input from a user when running an automated task.

Overview

Some tasks that are interactive, require the user to provide inputs. The Prompt command allows you to request inputs such as inputting values, opening a file/folder or/and choosing between yes and no whenever required while running such automated tasks.

Sub-Commands

The Prompt command provides the following sub-commands:

- Prompt for Value
- Prompt for File
- Prompt for Folder
- Prompt for Yes/No

Using the Sub-Commands
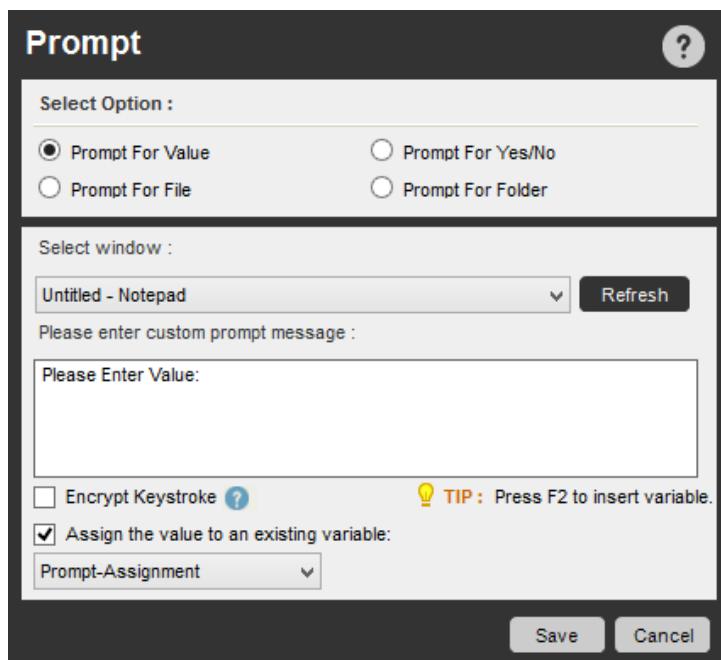
To insert a Prompt command in your task, follow these steps:

1. Double-click or drag one of the **Prompt** commands to the Task Actions List pane. The Prompt window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.

Prompt for Value Command

Use this command in tasks where the values change frequently. For example, for tasks that interact with online forms, you can ensure that the user enters the proper values when the task runs.

Further, the value that the user enters during task playback can be assigned to a variable. The variable can be used in other places within the task, instead of prompting again.

(Image as seen in Enterprise edition version 8.1)

You can also prompt users for passwords and encrypt the keystrokes during playback by enabling the 'Encrypt Keystrokes' option*. You will not be able to assign the value to a variable though, if it is enabled.
* Available from version 8.1 onward

(Image as seen in Enterprise edition version 8.1)

Prompt for File Command

Use this command to prompt the user for one or more files from a single folder. The value of the files selected can be assigned to a list variable. You can also configure a caption for the prompt window.

(Image as seen in Enterprise edition version 8.1)

Prompt for Folder Command
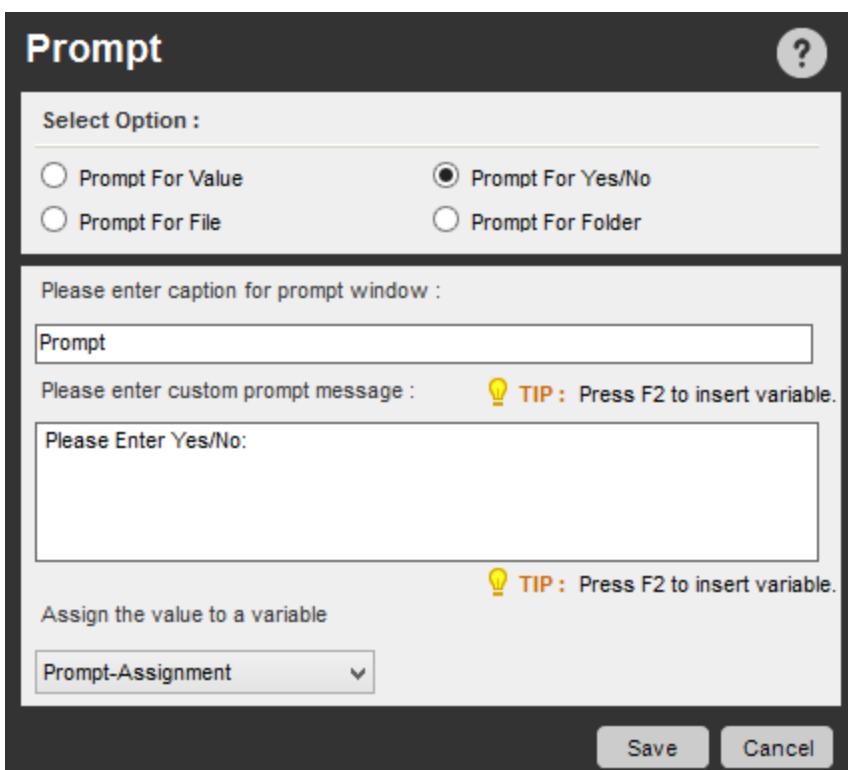
Use this command to prompt the user for a single folder. The value of the folder selected can be assigned to a value variable. You can configure a caption for the prompt window.

Prompt ?

Select Option :

○ Prompt For Value   ○ Prompt For Yes/No
○ Prompt For File   ● Prompt For Folder

Please enter caption for prompt window :

`Prompt`

Please enter custom prompt message :   💡 **TIP :** Press F2 to insert variable.

`Please Enter Folder Name:`

💡 **TIP :** Press F2 to insert variable.

Assign the value to a variable

`Prompt-Assignment ⌄`

Save   Cancel

(Image as seen in Enterprise edition version 8.1)

If Command

Use this command to prompt the user for a response (Yes or No) to a question while the task is running. The value selected during playback can be assigned to a value variable.

Prompt ?

Select Option :

○ Prompt For Value   ● Prompt For Yes/No
○ Prompt For File   ○ Prompt For Folder

Please enter caption for prompt window :

`Prompt`

Please enter custom prompt message :   💡 **TIP :** Press F2 to insert variable.

`Please Enter Yes/No:`

💡 **TIP :** Press F2 to insert variable.

Assign the value to a variable

`Prompt-Assignment ⌄`

Save   Cancel

(Image as seen in Enterprise edition version 8.1)

Also See...

- Prompt Command - Examples
- Message Box Command
- Variable Operation Command
- Knowing When to Use Variables

Read from CSV/Text Command
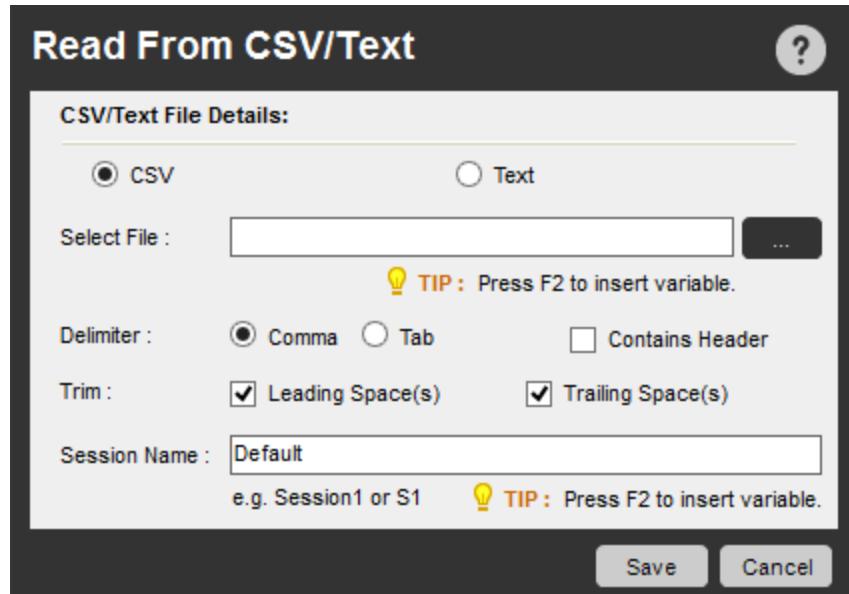
Overview

You can use this command to automate the process of reading data from various files and file formats. You can do the following:

- Read comma or tab-delimited data from a CSV file.
- Read multiple lines in CSV or text files
- Use encoding options: ANSI, UNICODE, and UTF8.
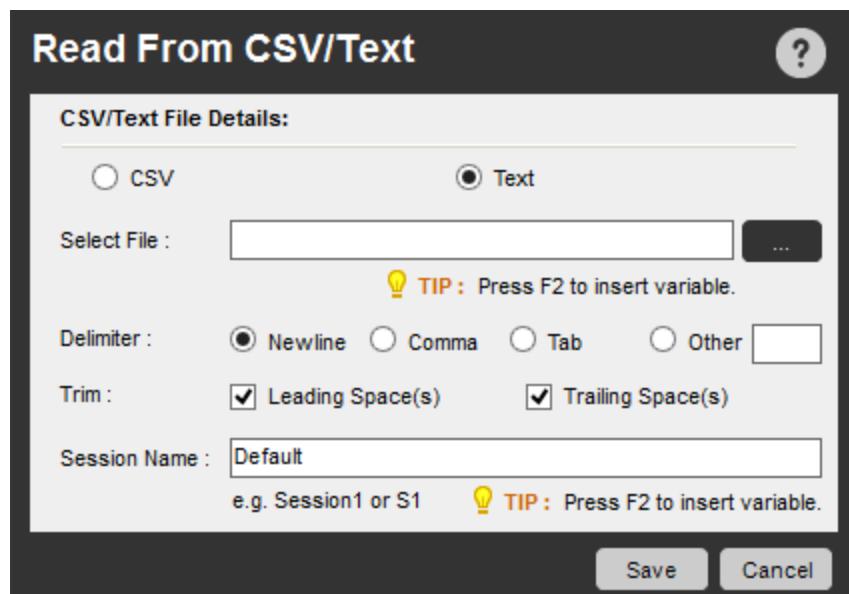
CSV

Use this option to read data using Comma or Tab delimiters from a CSV file.

Text

Using this command, read data using Newline, Comma, Tab or Other delimiters from a Text file.

Also See...

- Read from CSV/Text Command - Examples
- App Integration Command
- Database Command
- Email Automation Command
- Excel Command
- OCR Command
- PDF Integration Command
- Terminal Emulator Command
- XML Command
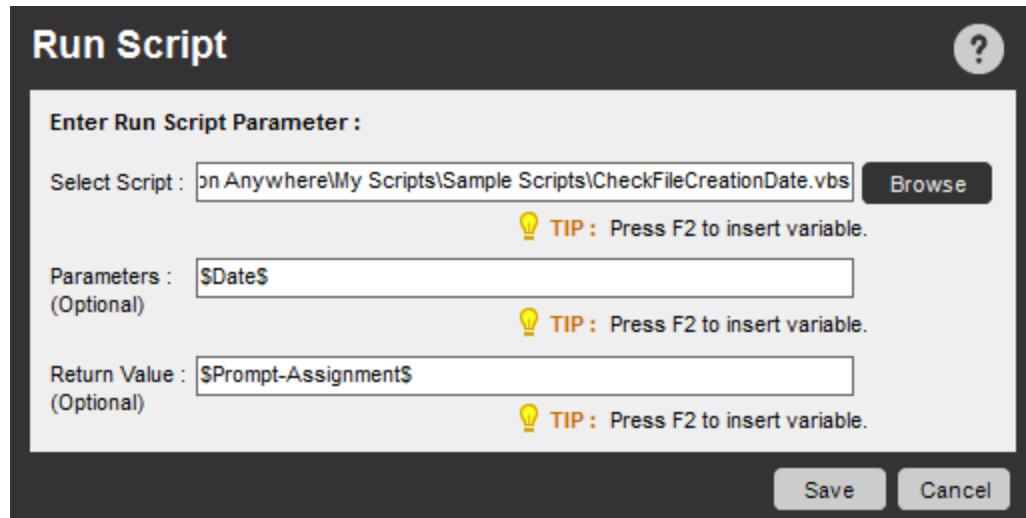
Run Script Command

Overview

You can use this command to run various scripts from your task.

Use a Run Script command when you want to:

- Run a VB Script (*.vbs) or JScript (*.js) in the task.
- Pass parameters to your scripts.
- Specify the Return Value in your scripts.

Run Script ❓

Enter Run Script Parameter :

Select Script : `on Anywhere\My Scripts\Sample Scripts\CheckFileCreationDate.vbs` Browse

💡 **TIP :** Press F2 to insert variable.

Parameters :
(Optional) `$Date$`

💡 **TIP :** Press F2 to insert variable.

Return Value :
(Optional) `$Prompt-Assignment$`

💡 **TIP :** Press F2 to insert variable.

Save   Cancel

Also See...

- Run Script Command - Examples
- Run Task Command
- Variable Operation Command

Run Task Command

Overview

Using the Run Task command, you can run several small tasks from within a larger task. You can create a single task, and within the task run other tasks, one after the other. This is called "*nesting tasks*".

Using this command, scheduling becomes easier. You can imbed several tasks into a single, master task, and simply schedule the master task. This enables you to avoid having to schedule each of the smaller tasks.
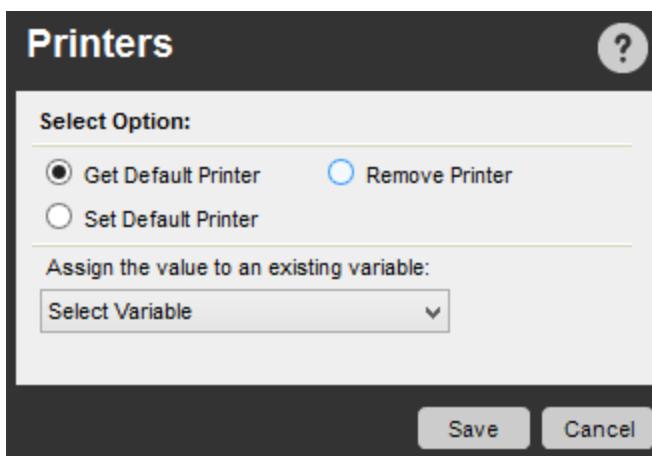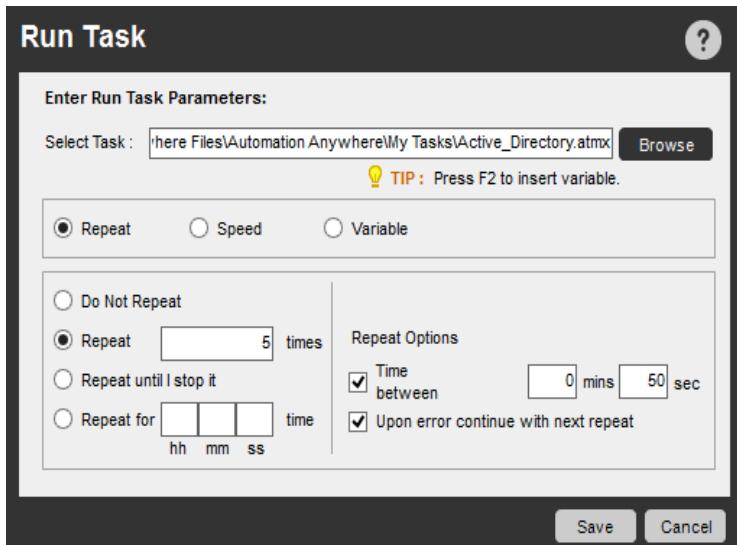
Using the Run Task Command

To insert a Run Task command in your task, follow these steps:
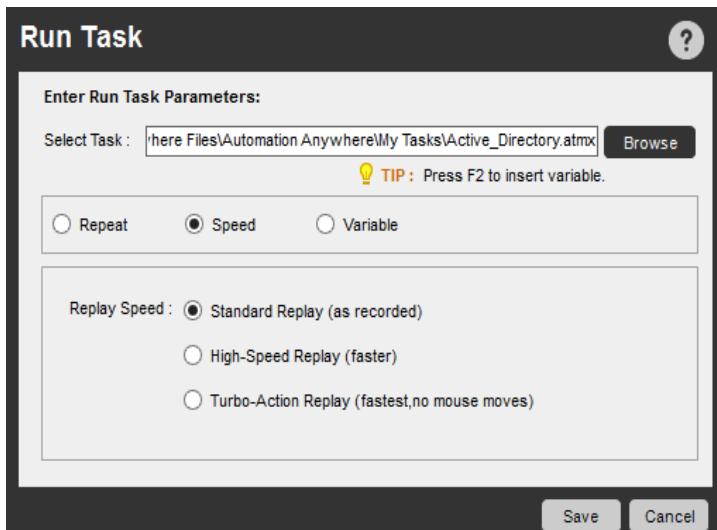
1. Double-click or drag the **Run Task** command to the Task Actions List pane. The Run Task window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.

The Run Task command includes the following options:

- **Repeat:** Use the Repeat option to run the selected task multiple times. You can repeat the task a specified number of times or for a number of hours. You can also repeat the task until you choose to stop it.

- **Speed:** Use the Speed option to set the replay speed of the task. Three speeds are available: Standard (same speed as recorded), High Speed, and Turbo Action replay (mouse moves are removed).

- **Variable:** Use this option to pass (associate) a variable from the master task - 'Main Task Variable' to the sub task - 'Run Task Variable'.
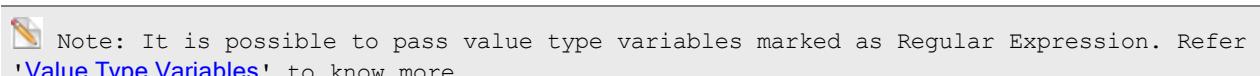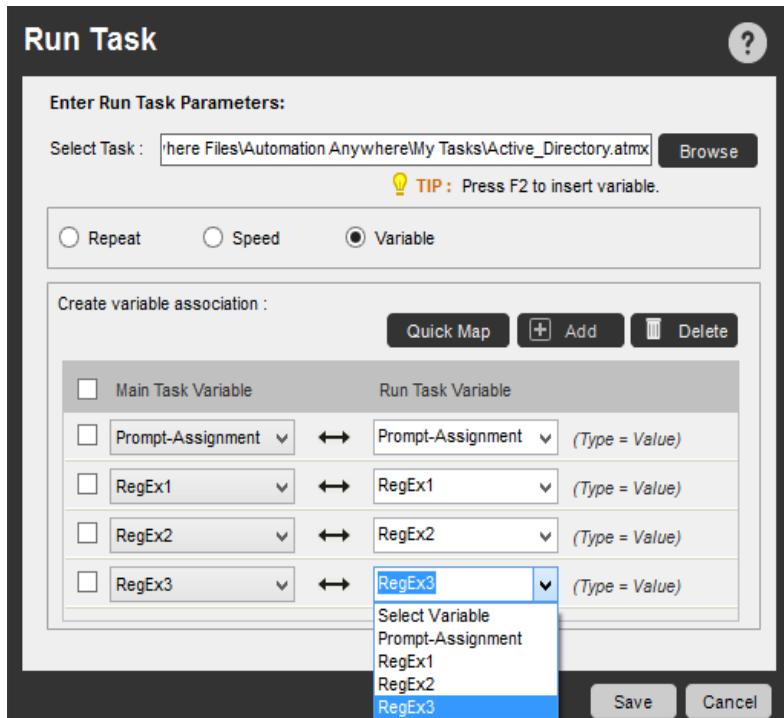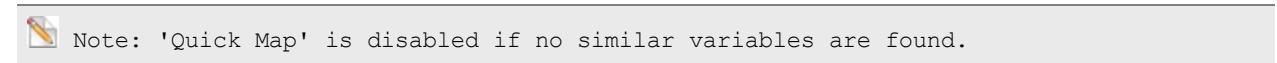
    Use the Add and/or Delete buttons to add and/or delete the variables.

    - **Inserting variables in 'Run Task Variable':** In order to specify the mapping variables for Run Task, you can either select a variable from the existing variables list or you can enter a new variable.

    - **Quick Mapping of Variables *:** A 'Quick Map' ensures that a large number of variables with same nomenclature are associated between tasks.

        * Available from version 7.5.2 onward.

    Variables mapped between tasks in earlier versions (below 7.5.2), will have to be re-mapped. Also, on selection of another task (using browse), the current mapping will be lost.

    Mapping of 'Workflow Variables' is also possible in similar fashion.

> 📝 Note: 'Quick Map' is disabled if no similar variables are found.

> 📝 Note: It is possible to pass value type variables marked as Regular Expression. Refer 'Value Type Variables' to know more.

Use Case Scenario

A task named `T1` has a variable *firstname* with a value of `'John'` declared. By using the Run Task command, another task, `T2`, is called and *firstname* is passed as an argument. Now, in task `T2`, the *firstname* value is changed to `'Michael'`. When the Run Task command finishes in task *T1*, the variable *firstname* contains the value `'Michael'`.

Also See...

- Run Task Command - Examples
- Run Script Command

- Stop Task Command
- Pause Command

Screen Capture Command

Use this command to automate the process of capturing screen shots.

Overview

As part of your automation task, you can capture the entire computer screen or an active open window.

Sub-Commands

The Screen Capture command provides the following sub-commands:

- Capture Desktop
- Capture Window
- Capture Area
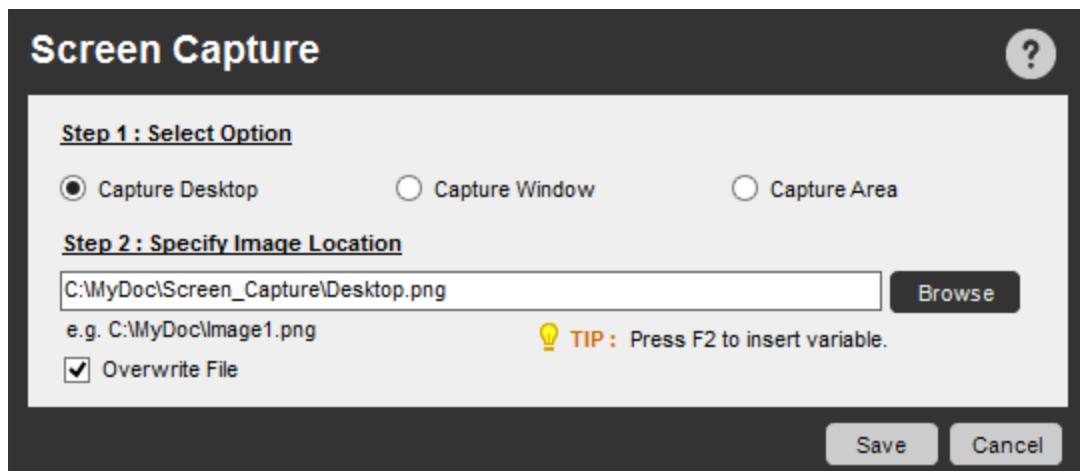
Using the Sub-Commands

To insert a Screen Capture command in your task, follow these steps:

1. Double-click or drag one of the **Screen Capture** commands to the Task Actions List pane. The Screen Capture window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.

Capture Desktop Command

Use this command to capture the full desktop on your computer. The image is saved to a location you specify. You can capture the image in various formats.

Capture Window Command

Use this command to capture an image of an open application window. Select the window by clicking on the name in the drop-down list.

The image is saved to a location you specify. You can capture the image in various formats.

Capture Area Command

Use this command to capture an area of the screen. Select the area by first selecting the application window, and then dragging the mouse over the area that you want to capture.

The image is saved to a location you specify. You can capture the image in various formats.

Use the Overwrite File option to overwrite older existing images.

Using Wild Cards

You can specify wild cards, using the asterisk (*) symbol, when selecting window titles. For example, you might use a wild card capture any window that contains the word "Untitled" in the title.

You might also want to capture any window that contains any text before or after the word "Untitled" in the window title. To specify this, use `*Untitled*` for the window title. In this case, Automation Anywhere searches for the exact window title ("Untitled"). If it is not found, Automation Anywhere searches for windows that contain the word "Untitled" in the title.

You can use a wild card character (*) at the beginning or at the end of a window title.

Also See...

- Screen Capture Command - Examples
- Clipboard Command
- Comment Command
- Error Handling Command
- Image Recognition Command
- Play Sound Command
- String Operation Command
- Variable Operation Command

Send Email Command

Overview

Automating email processes is one of the most powerful uses of Automation Anywhere. Sending repetitive emails can bog down the most productive organizations.

Using the Send Email command, you can:

- Send multiple emails at once. Use a semi-colon between each email address.
- Include attachments with the email.
- Format the message in HTML.

Using the Send Email Command

To send an email from your task, follow these steps:

1. Click on the Tools menu, and select Options. In the Email Settings, enter a Host Name and Port:
    - **Host:** The SMTP Host allows you to send email from Automation Anywhere. You can obtain this information from your email provider, or from the configuration settings in your email program.
    - **Port:** This setting defines the TCP/IP port that Automation Anywhere uses to send email. (The default port is 25.)

        Refer Setting Email Notifications for details

3. Double-click on or drag the Send Email command to the Task Actions List in the Task Editor. The Send Email window is displayed.

Send Email

Select Email Parameters :

From : `<from-email-address>`

TIP : Press F2 to insert variable.

To : `<to-email-address>`

TIP : Press F2 to insert variable.

Cc:

TIP : Press F2 to insert variable.

Bcc:

TIP : Press F2 to insert variable.

Subject: Automation Anywhere finished executing the task.

TIP : Press F2 to insert variable.

Attachment:

Message : ● Text ○ HTML   TIP : Press F2 to insert variable.

Message from Automation Anywhere.

Task Name : `<taskname>`

Description : `<desc>`

Repeat : `<repeat>`

Status: `<status>`

Last Run Time: `<lastruntime>`

TIP : Press F2 to insert variable.

Save   Cancel

(Image as seen in Enterprise edition version 8.1)

4. Enter the From, To, CC, and BCC email addresses.

5. Enter a subject.

6. Attach files you want to send with the email.

  - If you want to attach any file type, click the 'Attach' button.

  - If you want to attach values of all variables, click the 'Variables' button*.
    * Available from version 8.1 onward

This will append a text file that lists values of all the variables (System and User Defined) used in the task.

Attachment: `D:\AALog.txt<KB>; <AttachVariables>;`

(Image as seen in Enterprise edition version 8.1)

> Note: The System Variable *Excel Cell* will not be attached as the values input are dynamic.

7. Select either Text or HTML for email format.

8. Enter the message in the message box.

9. Save the command.

Also See...

- Send Email Command - Examples
- FTP-SFTP Command
- Internet Connection Command
- Launch Website Command
- Web Recorder Command
- Web Service Command

Services Command

Use this command to automate your Windows and application services.

Overview

You can use the Services command to start, stop, pause, resume, or get the status of any available services.

Sub-Commands

The Services command provides the following sub-commands:

- Get Service Status
- Start Service
- Stop Service
- Pause Service
- Resume Service
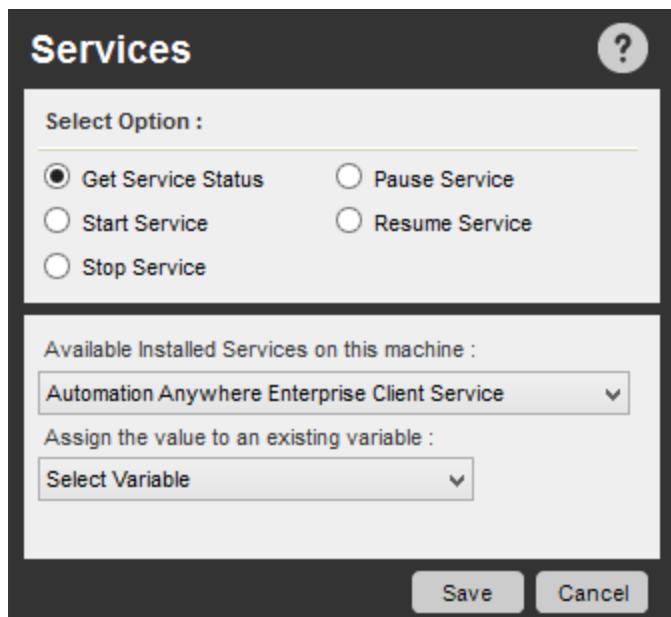
Using the Sub-Commands

To insert a Services command in your task, follow these steps:

1. Double-click or drag one of the **Services** commands to the Task Actions List pane. The Services window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.

Get Service Status Command

Use this command to obtain the current status of a Windows or application service. For example, the Windows Audio service status might be "running". You can assign the status value to a variable.

Start Service Command

Use this command to start a Windows or application service when your task runs.

Stop Service Command

Use this command to stop a Windows or application service that is currently running.

Pause Service Command

Use this command to pause a Windows or application service that is currently running.

Resume Service Command

Use this command to re-start a Windows or application service that you have previously paused using a Pause Service command.

Also See...

- Services Command - Examples
- Active Directory Command
- Printer Settings Command
- SNMP Command

SNMP Command

Use this command to automate the management of your network.

Overview

Simple Network Management Protocol (SNMP) is used to locate the network management component on one or more computers and the managed component on multiple network devices.

The Automation Anywhere SNMP command offers powerful network management. Using this command you can easily monitor network devices configured with SNMP agent software. Network devices such as servers, workstations, printers, routers, bridges, and hubs, as well as services such as Dynamic Host Configuration Protocol (DHCP) or Windows Internet Name Service (WINS) can be monitored.

Sub-Commands

The SNMP command provides the following sub-commands:

- Get
- Get Next
- Set
- Walk
- Send Trap

Using the Sub-Commands

To insert a Services command in your task, follow these steps:

1. Double-click or drag one of the **SNMP** commands to the Task Actions List pane. The SNMP window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.

SNMP Get Command

Use this command to retrieve data from an SNMP agent.

SNMP Set Command

Use this command to modify data on an SNMP agent.
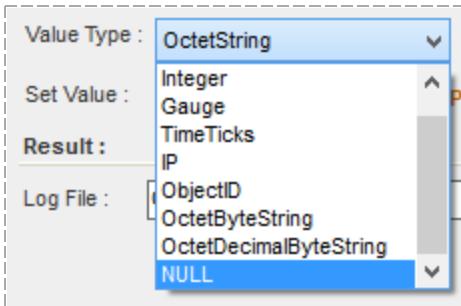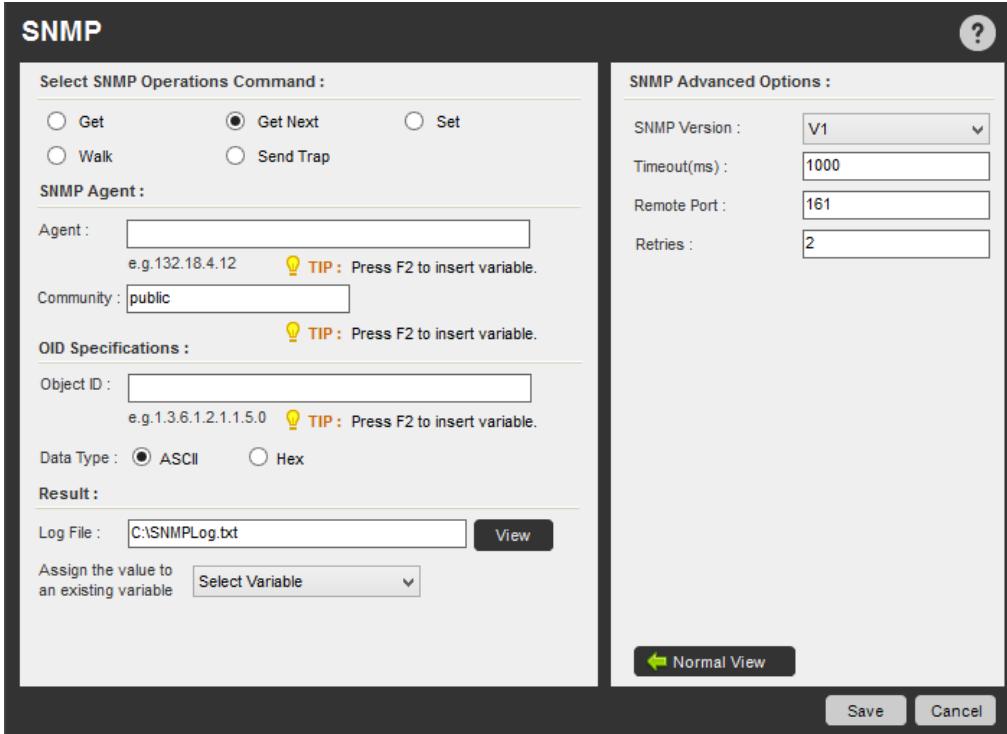
📝 Note: Select a 'Value Type' when providing the OID Specifications.

SNMP Get Next Command

Use this extended request command to browse the entire hierarchy of management objects.
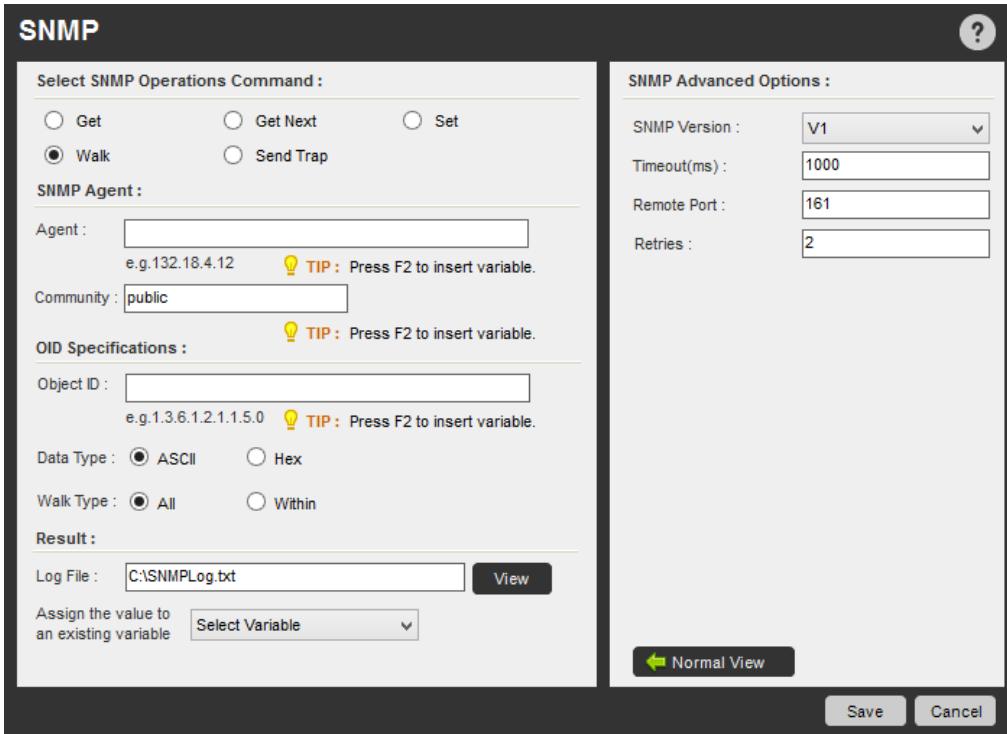
SNMP Walk Command

Use this command to retrieve a sub-tree of management values using SNMP GET-NEXT requests.

You can specify a Walk type of *All* or *Within*.
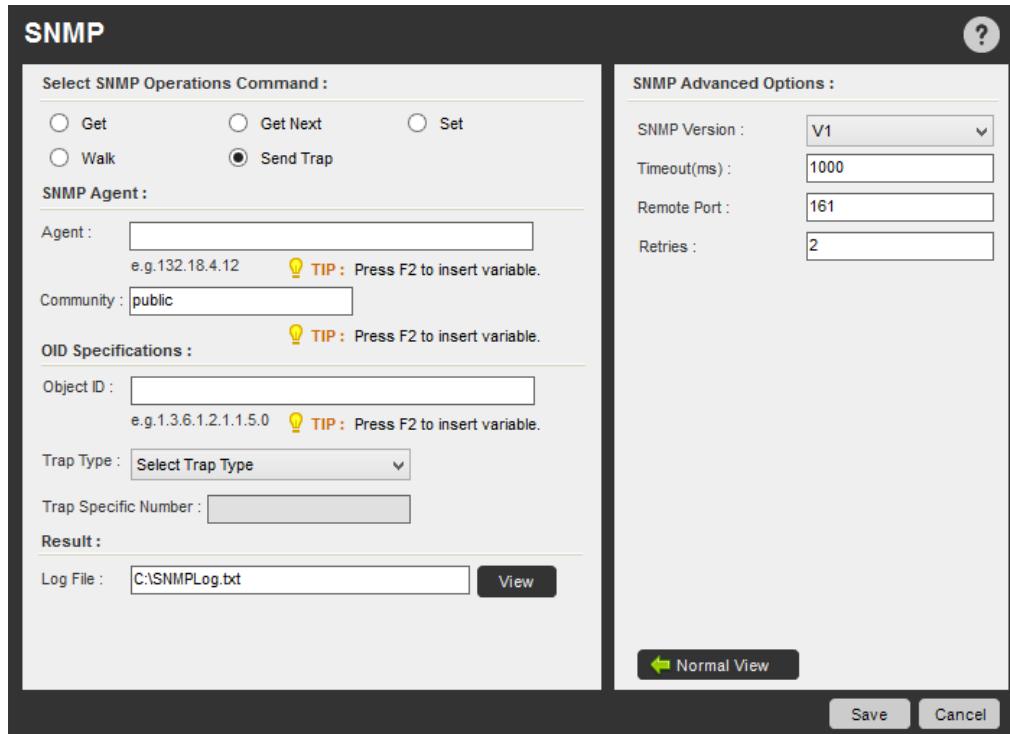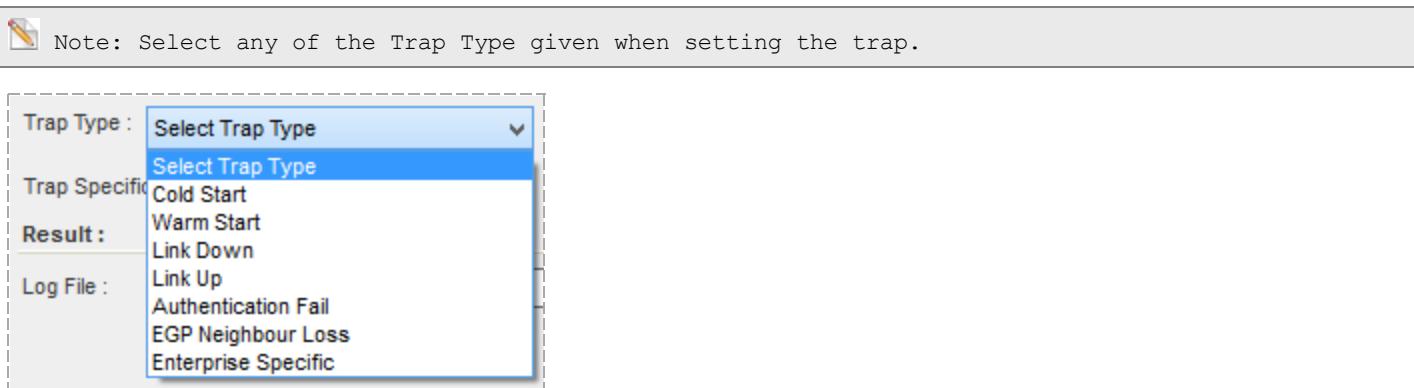
SNMP Trap Command

Use this command to send trap messages to the SNMP Manager from an SNMP Agent. These messages are sent by an SNMP Agent whenever certain events occur, such as a system restart.

> Note: Select any of the Trap Type given when setting the trap.

Using the SNMP Command Views

Using the Normal View

To use the normal SNMP command view, follow these steps:

1. **Select an SNMP Operations Command:** Select a command (Get, Get Next, Set, Walk, or Send Trap).
2. **SNMP Agent:** Specify an agent and community details.
   - **Agent:** Specify an IP Address or host name.
   - **Community:** Specify a community string. This string is used for read-only access to a network element.
3. **OID Specifications:** Specify the Object Identifier, which the SNMP Manager requests (for example, Get, Set, etc.)
   - **Object ID:** Specify the IP Address or host name.
   - **Data Type:** Select from ASCII or Hex.
   - **Set value:** Set value if the message type is Set.

- **Walk Type:** Select Walk Type of *All* or *Within* when the message type is Walk. For type Within, all variables in the sub-tree below the given OID are queried. Using All displays the variables that are equal to the specified OID for the given tree type structure.

- **Trap type:** Select a trap type from the following:

    a. **Cold start:** Occurs when the SNMP agent initializes its configuration tables.

    b. **Warm start:** Occurs when the SNMP agent re-initializes its configuration tables.

    c. **Link Down:** Occurs when the state of a network adapter on the SNMP agent changes from up to down.

    d. **Link Up:** Occurs when the state of a network adapter on SNMP agent changes from down to up.

    e. **Authentication Fail:** Occurs when the SNMP agent receives a message from an SNMP manager with an invalid community name.

    f. **EGP Neighbor loss:** Occurs when the SNMP agent cannot communicate with its Exterior Gateway Protocol (EGP) peer.

    g. **Enterprise specific:** Occurs when specific error conditions and error codes are defined in the system. The user must provide a Trap Specific Number for this option.

4. **Result:**

    - **Log File:** View the SNMP log data in the selected log file.

    - **Assign the value to an existing variable:** Specify a variable value.

Using the Advanced View

When you click on the Advanced View button, additional SNMP options are provided. The SNMP advanced options include the following:

- **SNMP Version:** Select any of device versions - V1, V2 or V3

- **Timeout(ms):** Specify the response time in milliseconds for the action to wait for before generating an error.

- **Remote Port:** Specify the TCP/IP port to be used for the Get request. The default is 161.

- **Retries:** Specify the number of times to retry the message.

V3 allows you to select authentication levels to ensure a more secure environment (compared to V1 and V2). The options provided therein are:

- **Authentication Level:** Choose from 'None', 'Authentication Only' or 'Authentication with privacy'.

- **Authentication Method:** Select either MD5 or SHA.

> 📝 Note: Applicable only if Authentication Level selected is other than 'None'. You are also required to input a 'Password'.

- **Password:** Input a password for connecting to the device.

- **Privacy Method:** Select the either 'DES' or 'AES' as the privacy method. This privacy method ensures that your password is protected with a private key as the program encrypts the same.

> 📝 Note: Available only if authentication level selected is 'Authentication with privacy'.

- **Private Key:** Input the private key generated when the authentication level selected is 'Authentication with privacy'.

Also See...

- SNMP Command - Examples
- Active Directory Command
- Services Command
- Printer Settings Command

Stop Task Command

Use this command to stop an automation task from running.

Overview

You can stop an automation task at any point by inserting a Stop Task command in the Task Actions. For example, you can stop the task if a specific condition is met, such as when encountering a file larger than 100 MB.

Using the Stop Task Command

To insert a Prompt command in your task, follow these steps:

1. Open the task in the Task Editor.
2. Double-click or drag the **Stop Task** command to the Task Actions List pane.
3. To change the location of the Stop Task command, simply cut/paste it or drag it to another location within the automation task.

Automation Anywhere inserts the Stop Task command above the currently selected line in the Task Actions List.

 Stop The Current Task

Also See...

- Run Task Command
- Run Script Command

String Operation Command

Use this command to manipulate a text string or extract part of a string and store it in a variable.

Overview

String operations are very useful when automating business processes. For example, you might want to extract a portion of a text block from an email, a website, or an application window.

You can also manipulate strings after capturing them. You might search the string for particular phrases, compare two strings, convert a string to upper or lowercase, or obtain the length of a string.

Sub-Commands

The String Operation command provides the following sub-commands:

- Before/After
- Compare
- Find
- Join
- Length
- Lower Case
- Replace
- Reverse
- Split
- Sub-String
- Trim
- Upper Case

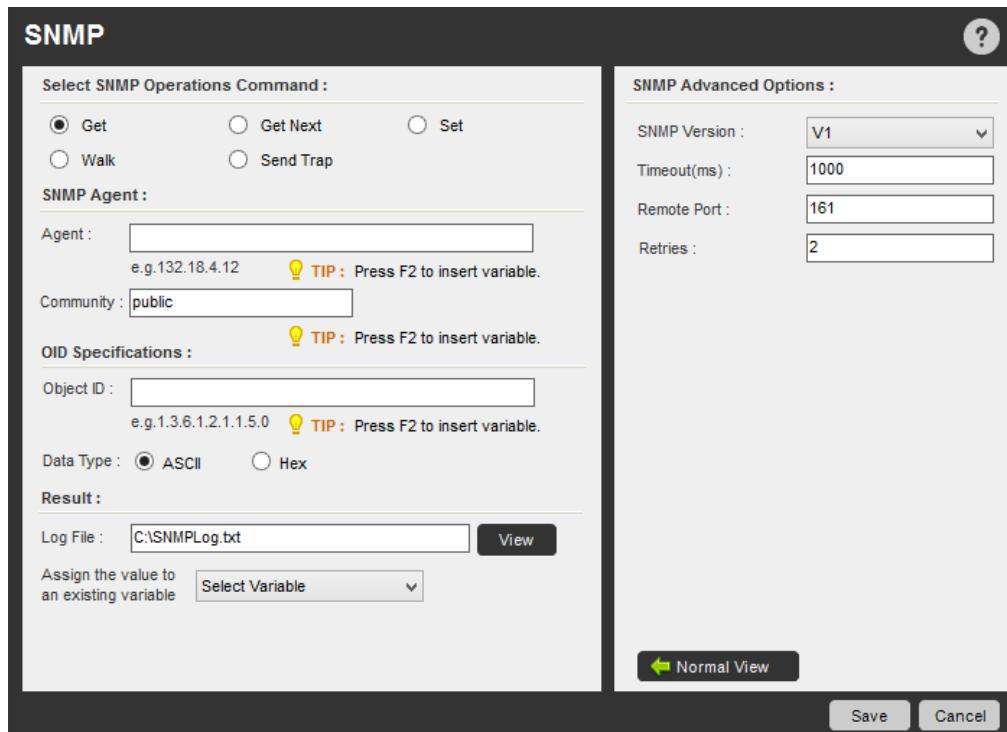## Using the Sub-Commands

To insert a String Operation command in your task, follow these steps:

1. Double-click or drag one of the **String Operation** commands to the Task Actions List pane. The String Operation window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.

Before/After Command

Use this command to specify a range of text to extract, using **Before** and **After** keywords.

You can insert 'Enter' and 'Tab' variables by using the F2 function key. These variables act as separators for your string between the **Before** and **After** keywords.

To refine your 'Before-After' string command, you can use the 'OR' or 'AND' Logical Operators.

- Use the 'AND' operator to ensure 'Before' and the 'After' conditions are met.
  - For instance, when you input a string in 'Before' and if not found, the search will be aborted. The program will not search the string input in 'After'.
- Use the 'OR' operator to ensure either 'Before' or 'After' condition is met.
  - For instance, when you input a string in 'Before' and if not found, the program will continue to search for the string input in 'After'.

> Note: The default Logical Operator is 'OR'

String Operation

Select Operation : Before-After

Note : Returns the sub string enclosed by 'Before' and 'After' strings.

Source String :

TIP : Press F2 to insert variable.

Before (Optional)

Occurrence(Optional) 1

TIP : Press F2 to insert variable. TIP : Press F2 to insert variable.

Logical Operator OR

After (Optional)

Occurrence(Optional) 1

TIP : Press F2 to insert variable. TIP : Press F2 to insert variable.

Example :
To extract $249 from 'Price: $249 xyz' , Specify "Before" = Price: _"After" = xyz

No. of characters to be extracted

☑ Trim the extracted text ☐ Remove Enter from the extracted text

If no match found, return ⦿ Source String ◯ Empty (Null) String

Assign the output to variable : Select Variable

Save Cancel

Example: In the text string, *'Name of Applicant James Smith Applicant Location,'* let's copy only *'James Smith'*. We specify **Before** = *'Applicant'* with an Occurrence of 1. We also specify **After** = *'Applicant'* with an Occurrence of 1. This is the first occurrence after the **Before** keyword, although it is the second occurrence from the beginning of the string.

You can also specify the number of characters to be extracted, starting from the first character after the **Before** keyword. The extracted string can be trimmed to remove leading or trailing spaces, and you can save the string to a variable.

> 💡Tip:
> - When using the **Before/After** command, enter a space as a separator between strings using the SPACE BAR key on the keyboard.
> - In some cases, content extracted from websites are separated by an Enter symbol (a small square), which you can see in the CSV file.

Compare Command

Use this command to compare strings by specifying the strings. You can refine your search by selecting the 'Match Case' option. A value of True or False is returned. You can assign the values that are returned to a variable.

Find Command

Use this command to locate a sub-string that exists within the source string by specifying the Start parameter.

If the specified string is a regular expression, select the check box that is provided. Insert the regular expression in the **Find String** field. Also, refine your search parameters by specifying the character number in 'Start From', selecting 'Match Case' and/or Regular Expression.

For example, to find all email addresses in the source, specify the following as a regular expression: `\b[A-Z0-9._%-]+@[A-Z0-9.-]+\.[A-Z]{2,4}\b`

Join Command

Use this command to join all separate elements of list variables by defining the source string and a delimiter.

Length Command

Use this command to obtain the length of a string by specifying the source string.

Lower Case Command

Use this command to convert a source string to lower case .

Replace Command

Use this command to replace a portion of a source string with a specified replacement string. Optionally specify the character position from which to begin in 'Start' and the number of times it is to be replaced in 'Count'. If the Find String field contains a regular expression, select the Regular Expression check box.
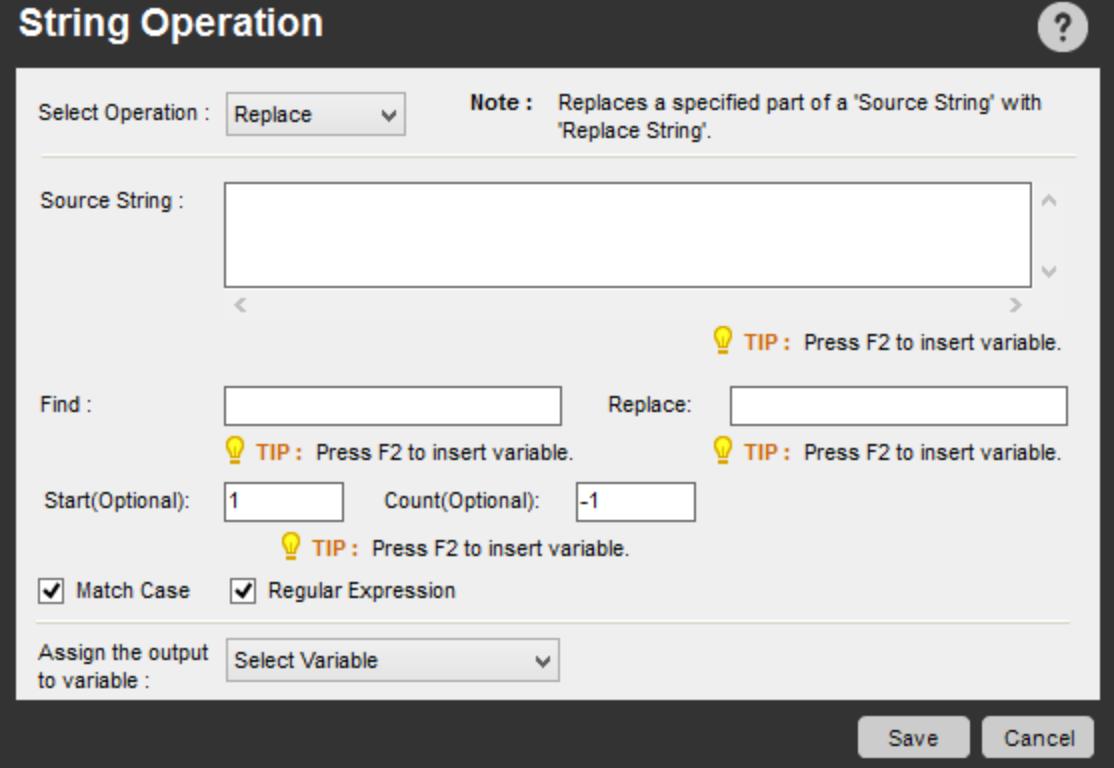
Reverse Command

Use this command to reverse a specified source string.

Split Command

Use this command to split a source string. Specify a 'Delimiter' and optionally number of delimiters to be considered in 'Limit'. Refine the scope by selecting 'Match Case'. The resulting elements are stored in a list variable.

Sub String Command

Use this command to retrieve a sub-string. Specify the source string and parameters such as 'Start From' and optionally 'Length'.

Trim Command

Use this command to trim blanks and spaces from a specified source string.

Upper Case Command

Use this command to convert a source string to upper case.

Also See...

- String Operation Command - Examples
- Variable Operation Command

System Command

Use this command to automate repetitive tasks pertaining to your computer system.

Overview

Certain repetitive and routine tasks pertaining to your computer system can be automated to save on time and consequently be conserve energy. This topic describes how to use such commands in Automation Anywhere.
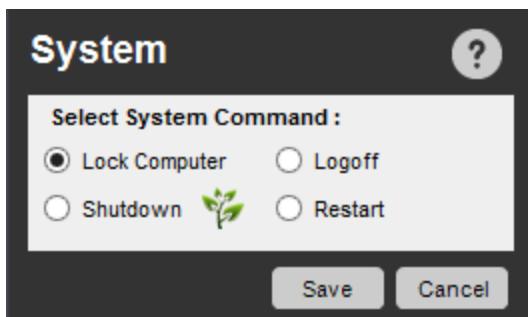
Sub-Commands

- Lock the computer
- Log off
- Shutdown the computer
- Restart the computer

Using the Sub-Commands

To insert a System command in your task, follow these steps:

1. Double-click or drag one of the **System** commands to the Task Actions List pane. The System window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.

Lock the Computer Command

Use this command to automatically lock the computer. This command is commonly used at the end of a task, to ensure security if no one is at the system.

Logoff Command

Use this command to automatically log off the computer. This command is commonly used at the end of a task, to ensure security if no one is at the system.

Shutdown Command

Use this command to turn off your computer, for example, at the end of the workday.

- The Shutdown command provides the added benefit of energy savings.
- To increase environmental awareness, Automation Anywhere highlights energy-saving actions with a green leaf icon, as shown in the System window next to the Shutdown command.

Note: The Shutdown command should only be inserted at the end of an automation task. Otherwise, other commands in the task cannot be run.

Restart Command

Use this command to automatically restart the computer.

> 📝 Note: The Restart command should only be inserted at the end of an automation task. Otherwise, other commands in the task cannot be run.

Also See...

- System Command - Examples
- File and Folder Commands
- Log-to-File Command
- Manage Windows Controls Command
- Open Program/File Command
- Windows Actions Command

Terminal Emulator Command

Available in Enterprise and Small Business editions only.

Use the Terminal Emulator (TE) command to access and control default, DOS-based actions of remote terminals

Overview

Access and control default, DOS-based actions of remote terminals with Terminal Emulator command. Issue DOS commands - `dir`, `ipconfig`, `copy`, and `recover` etc. Also, transfer the exact state of an application between tasks to optimize your automation logic.

Sub-Commands

The Terminal Emulator command provides the following sub-commands:

- Connect
- Disconnect
- Send Text
- Get Text
- Set Field
- Get Field
- Get All Fields
- Wait
- Control

Using the Sub-Commands

To insert a Terminal Emulator command in your task, follow these steps:
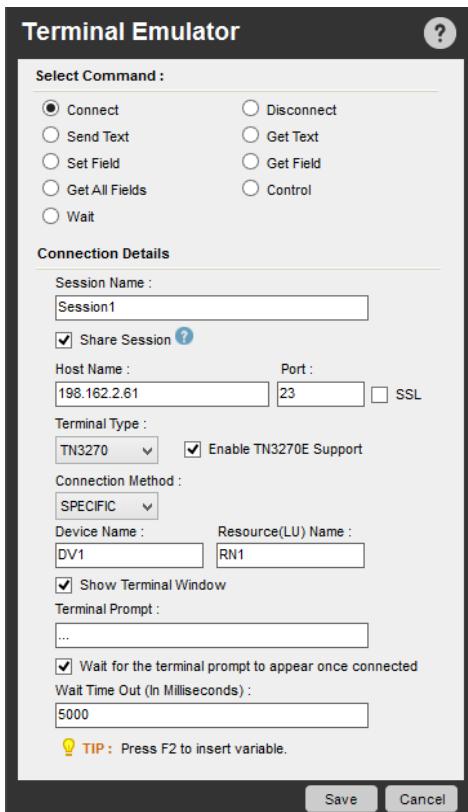
1. Double-click or drag one of the **Terminal Emulator** commands to the Task Actions List pane.
2. Specify required parameters in the fields - Session Name, Host Name, Port, Terminal Type and Terminal Prompt.
3. Click Save.

Connect Command

Use this command to remotely connect to a terminal. Specify the connection details of the terminal. You can connect to the following types of terminals:

- ANSI
- VT100
- TN3270
- TN5250

> Note: The default terminal selected is ANSI.

(Image as seen in Enterprise Edition 8.0)

Share Session*

* Available from version 7.5.2 onward

Enable the 'Share Session' option to optimize your automation logic. With it you can trim your bulky tasks as it provides the required flexibility to transfer the "exact state of the application to another task". You can use the Run Task command to achieve this.

> 📝 Note: Only Terminal Emulator command session state can be shared between tasks. The existing state is passed on to all Terminal Emulator sub-commands in the subsequent tasks (sub-tasks).

Reference Share Session between Tasks to understand how this option helps you achieve more.

TN Terminal - Options

Enabling Secure Connection

Enabling/Selecting 'SSL' ensures a secure connection for the Terminal Types 'TN3270' and 'TN5250'; the default 'Port' for the same being '992'.

- **Terminal Type TN3270:** Selecting 'Enable TN3270 Support' allows you to choose a 'Connection Method', which could be either 'Generic' or 'Specific'. You can hence:
  a. Input the 'Device Name' when you select a 'Generic' connection method.
  b. Input the 'Device Name' and 'Resource (LU) Name' when you select a 'Specific' connection method.

> 📝 Note: On enabling **TN5250 Support,** only the 'Device Name' and 'Resource (LU) Name' can be input.

Advanced View

Advanced view is available on selection of terminal types ANSI and VT100. Specify either of the three protocols viz. Telnet, SSH1, and SSH2 .

(Image as seen in Enterprise Edition 8.0)

> 📝 Note: The default connection type is Telnet.
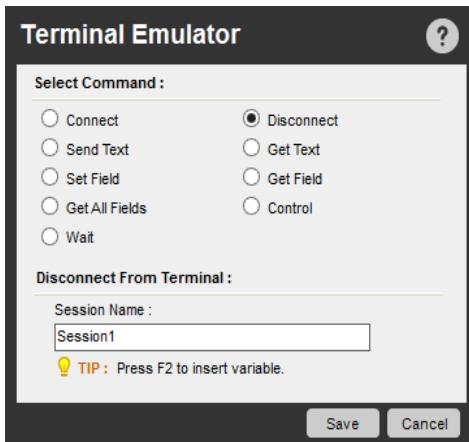
For **SSH1** and **SSH2**, provide an authentication type to connect to the terminal.

Authentication modes available:

- **User Authentication:** Specify a user name and password.`
- **Key file Authentication:** Select a location for the key file.

Disconnect Command
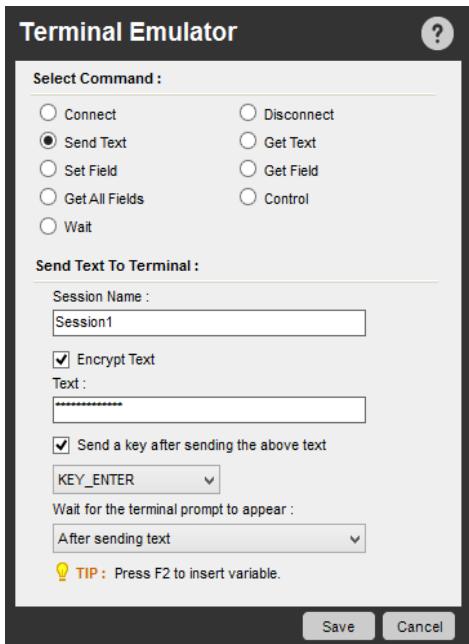
Use this command to disconnect from an existing terminal connection.

Send Text Command

Use this command to specify text and send that text to the terminal. You can also send encrypted text by selecting the Encrypt Text check box.

To send a key after sending the encrypted text, select the required key from the drop-down list.

Get Text Command

Use this command to receive text from the terminal. You can select from the following options:

- Last Line
- All Lines
- Line From-To
- Lines With Column Range

Set Field

Use this command to set field either by Index or Name for the terminal types TN3270 and TN5250. Setting data through field for such type of terminals enhances automation for them.
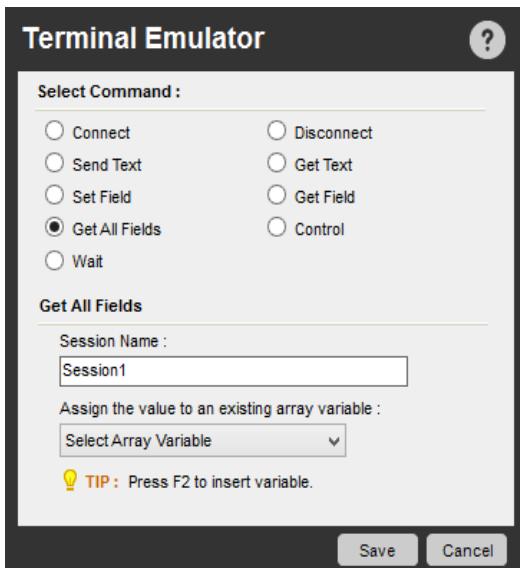
Get Field

Use this command to get a field value by Index or Name and assign it to a variable for the terminal types TN3270 and TN5250.
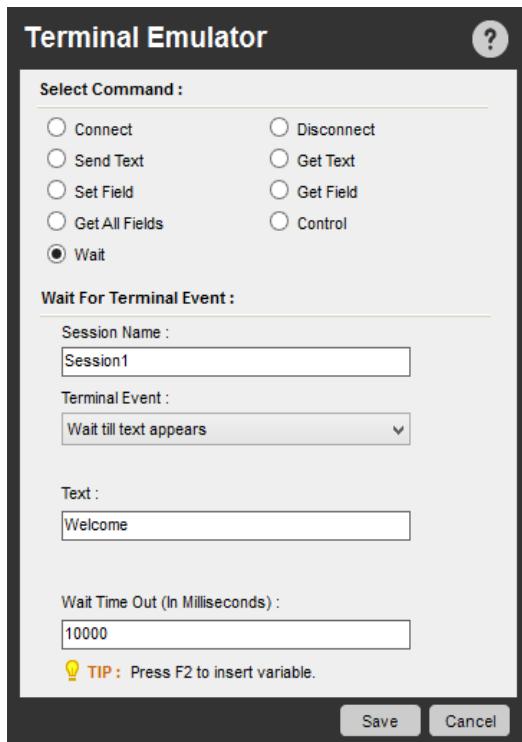
Get All Fields

Use this command to get value of all fields and assign them to array variables for the terminal types TN3270 and TN5250.

Wait Command

Use this command to insert a Wait command. Inserting a Wait command helps you to automate and trigger another action in your automated task based on the terminal status.
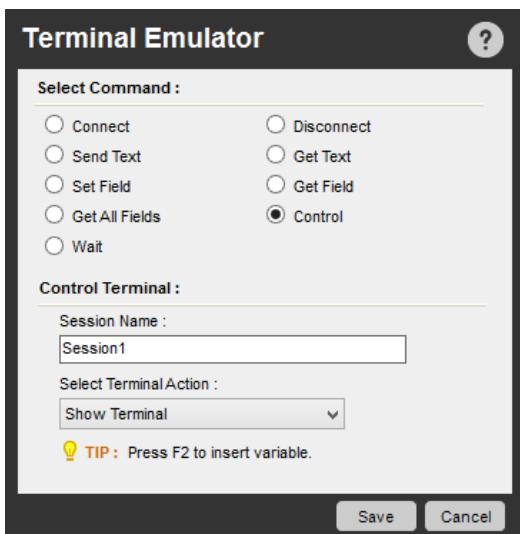
The following options are provided:

- Wait till text appears
- Wait till text disappears
- Wait till cursor moves to position
- Wait till cursor moves out of position
- Wait till screen gets blank
- Wait till screen contain text
- Wait till terminal prompt appears
- Wait till terminal Ready State

Control Command

Use this command to control how the terminal is viewed.

The following options are provided:

- Show Terminal
- Hide Terminal
- Clear Terminal
- Set Cursor Position

Also See...

- Terminal Emulator Command - Examples
- App Integration Command
- Database Command
- Email Automation Command
- Excel Command
- OCR Command
- PDF Integration Command
- Read from CSV and Text Files Command
- XML Command

NEW! Share Session Between Tasks

Available in Enterprise and Small Business editions only.

Imagine having to run hundreds of tasks concurrently, which use commands that are repetitive. For instance, having to connect to the same application, time and again, from different tasks can be cumbersome. An option that gives you the freedom to achieve this using a single task would help you optimize your automation logic to a greater extent, in the long run.

The 'Share Session' option is designed to achieve this, precisely. You can use the option 'Share Session' in Terminal Emulator command when you want to pass on the exact state of the application to the concurrently occurring tasks and back.

How to 'Share Session'?

You can create a hierarchy of tasks connected by a common thread - 'Share Session':

- Simply create a 'Main Task' that would connect you to the application with 'Share Session' option enabled/selected, using the Run Task command
- Call in other 'Sub Tasks' that obtain the necessary data/information.

In fact, you do not disconnect from the application (terminal in this case) in the Sub Tasks; you simply pass on the exact state of the application from Main Task to the Sub Tasks and back. This will ensure,

- Your bulky tasks are trimmed off of repetitive commands for connecting and disconnecting.
- Which in turn results in you achieving the same results, faster.

Example: Share session to access and fetch data from applications installed on a mainframe terminal
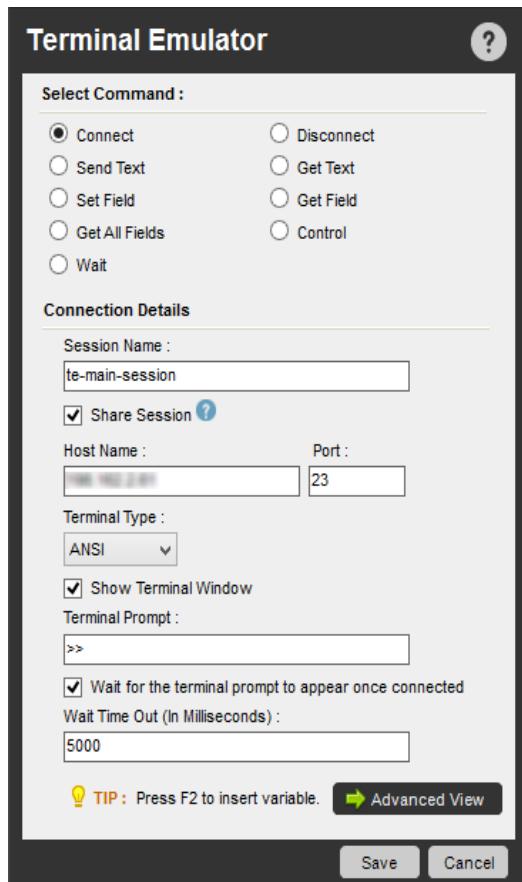
Here, we cite a hypothetical example to guide you through the workings of 'Share Session':

The assumption is that you have access to a mainframe terminal with various applications installed. Usually you would access each application using separate sessions to connect to the terminal. Hence, each task comprises of repetitive commands such as connect and disconnect (to clear the earlier data session).

Now that you are armed with the powerful option 'Share Session' you can literally share your existing session state between tasks.

Ideally you could create a Main Task - TE_Share_Main for instance, that calls the other Sub Tasks - TE_BMIS and TE_CMIS. These Sub Tasks obtain required data/information from various applications.
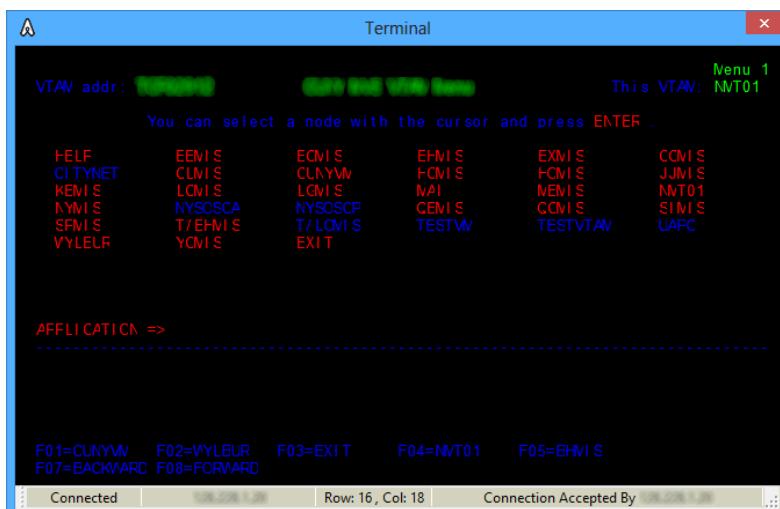
1. In the Main Task - TE_Share_Main, connect to the mainframe terminal using the 'Connect' sub-command in Terminal Emulator. Enable 'Share Session'.
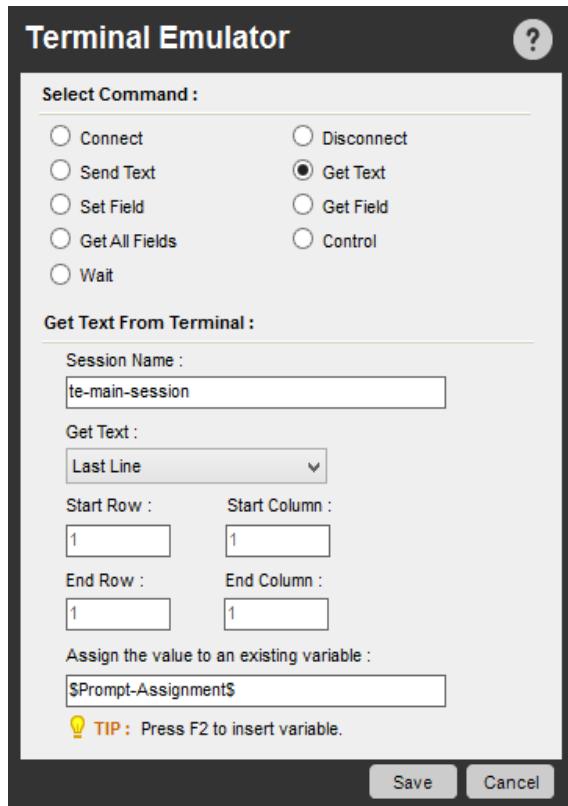
2. Include a 'Run Task' command to call in the first Sub Task - TE_BMIS. Ensure that this task has used some set of sub-commands such as 'Send Text' and 'Set Fields'.
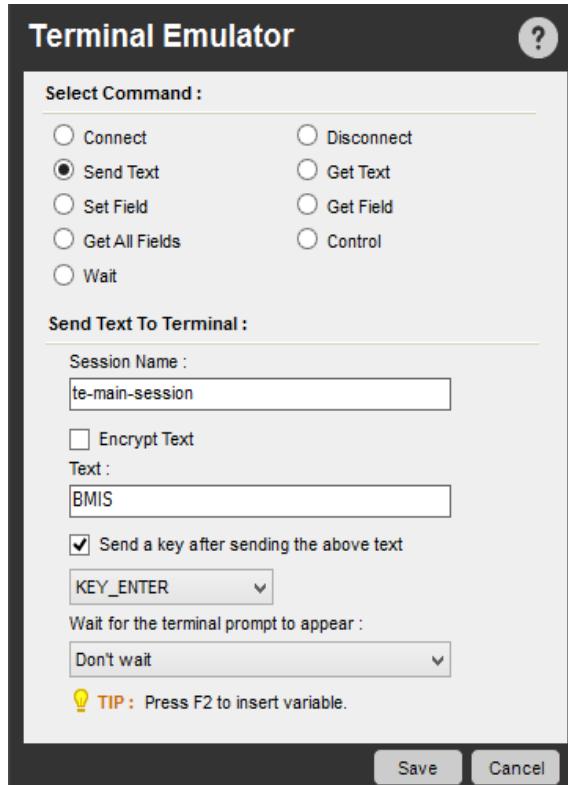
> Note: Since share session is enabled, the application BMIS starts in the same terminal that is connected in the Main Task - TE_Share_Main.
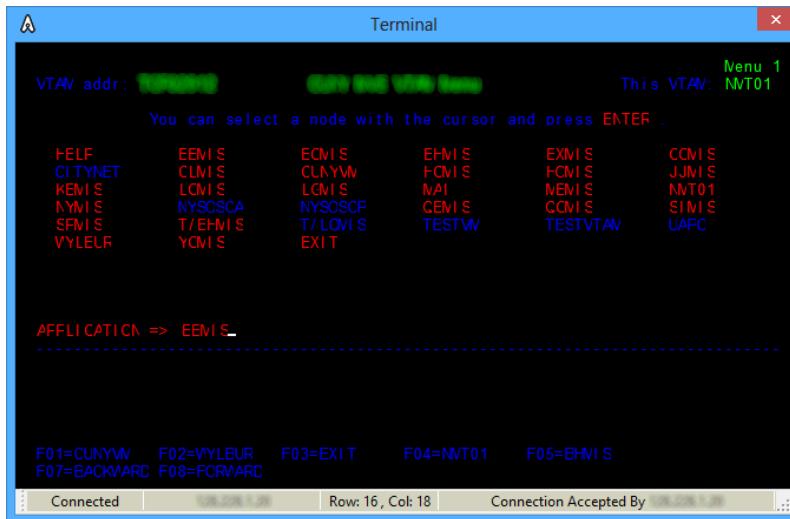
3. Use the 'Get Text' and 'Get Field' command to obtain the required information.
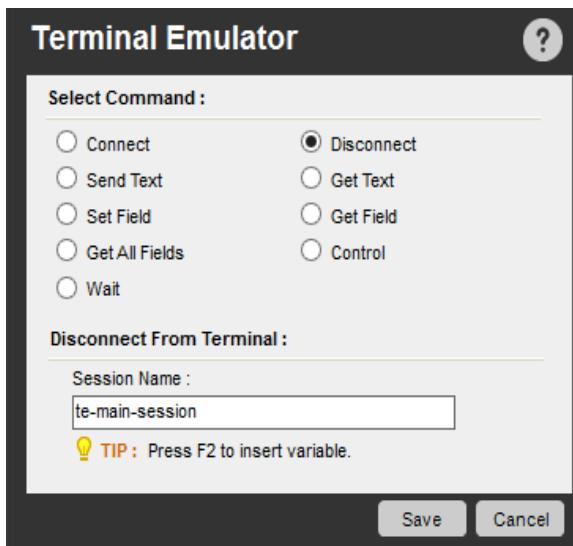
4. To restore the original terminal state, use the 'Send Text' command.

Other User
Resources:

User forums

Ask the Expert

Open a support case

Automation Examples

5. The Sub Task ends here and the control is passed on to the Main Task.

- For the next Sub Task, TE_CMIS, use similar steps - Connect with 'Share Session' option, then use 'Send Text' and 'Set Field' sets of sub-commands and acquire information using 'Get Text' and 'Get Field' set of sub-commands. Use the 'Send Text' command to return to Main Task and restore the terminal to it's original state.

6. Create as many Sub Tasks as required and return to the Main Task using Share Session!

7. End the session in the Main Task by adding a 'Disconnect' sub-command.

Variable Operation Command

Use this command to assign variables to a task.

Overview

The Variable Operation command has been designed for allowing you to assign or reinitialize user specified variables (user variables)

The type and source of variables used dictate the assignment and reinitializing. These are applicable during task execution thus allowing the user to reuse the user variables.

Using the Variable Operation Command

To use the Variable Operation command, follow these steps:
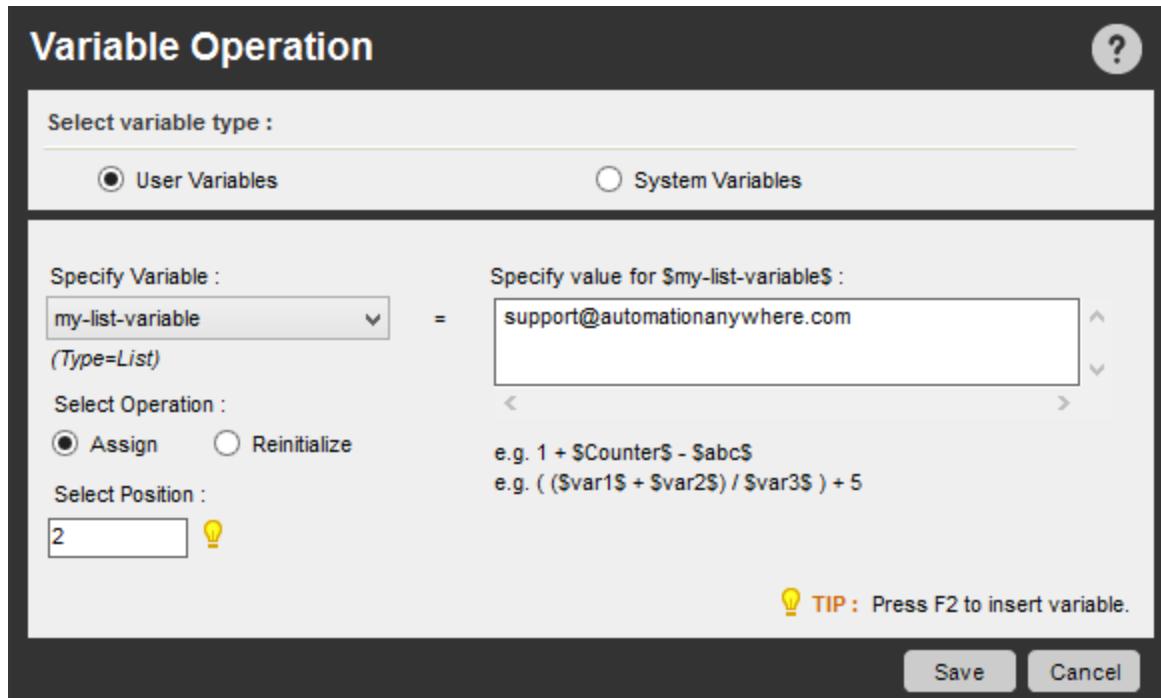
1. Select either User Variables or System Variables.

- **For User Variables:**
    a. Select a user variable to specify a value to an existing user variable that you have defined.
    b. Assign a value. You can assign a value to the variable. The right side of the 'equal to' operator can be a regular operation; for example, you can specify: `$var1$ = ($var2$ + $var5$ - 10) / 5`.

    > 💡 Note: The operators (,), /, *, + and - are supported. The left and right parentheses take precedence, and the operation within them is evaluated from left to right.

    Example: In the expression `($Var1$ + 5) * ($Var2$ -10)`, the first calculation `($Var1$ + 5)` will be evaluated first, then `($Var2$ -10)`, and the result of both will be multiplied and assigned to the variable specified on the left side of the equals sign.

Variable Operation

Select variable type :

- ⦿ User Variables
- ○ System Variables

Specify Variable :

my-list-variable ⌄   =

(Type=List)

Specify value for $my-list-variable$:

support@automationanywhere.com

Select Operation :
- ⦿ Assign   ○ Reinitialize

e.g. 1 + $Counter$ - abc
e.g. (($var1$ + $var2$) / $var3$) + 5

Select Position :

2   💡

💡 TIP : Press F2 to insert variable.

Save   Cancel

- **For System Variables:** This option enables you to reset one of the following system variables:
    a. Error Description
    b. Error Line Number
    c. Email From
    d. Email To
    e. Email Cc

f. Email Subject

g. Email Message

Also See...

- Variable Operation Command - Examples
- Using the Variable Manager
- Using Variables
- User (Local) Variables
- System Variables
- Comment Command
- Error Handling Command
- Image Recognition Command
- Play Sound Command
- Screen Capture Command
- String Operation Command

Web Recorder Command

Use this command to edit and enhance an automation task that you build using the Web Recorder.

Overview

The Web Recorder helps you to automate Internet-related task actions. Using the Web Recorder command, you can record tasks to extract data from the web, download data, test web sites, extract source code, and more.

Sub-Commands

The Web Recorder command provides the following sub-commands:

- Open and close websites in browser sessions
- Manage web controls
- Download data
- Extract additional data from the web
- Test websites and locate broken links
- Navigate URLs
- Extract Website source code
- Search by captions
- Navigate back
- Execute a Java Script

Using the Sub-Commands

To insert a Web Recorder command in your task, follow these steps:

1. Double-click or drag one of the **Web Recorder** commands to the Task Actions List pane. The Web Recorder window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.

For more information on using the specific sub-commands, see Editing a Web-only Task with Web Recorder Commands.

Web Recorder

Select Command :

- ◉ Open Browser
- ○ Manage Web Controls
- ○ Extract Table
- ○ Extract Data
- ○ Extract Source
- ○ Go Back

- ○ Close Browser
- ○ Download File
- ○ Find Broken Links
- ○ Navigate URL
- ○ Search By Caption
- ○ Execute JavaScript Function

Step 1: URL to Open

Step 2: *(Run this command in the existing IE Window with below properties.)*

☑ Page Title   Select Page Title / Type New   Refresh

☑ Page URL

💡 **TIP :** Press F2 to insert variable.

Save   Cancel

Note: It is recommended that you first search in the following sequence: By Caption, By Attributes and By Index.

Also See...

- Editing a Task with Web Recorder Command
- FTP-SFTP Command
- Internet Connection Command
- Launch Website Command
- Send Email Command
- Web Service Command

Web Service Command

Available in Enterprise and Small Business editions only.

Use the Web Service command to implement SOA (Service Oriented Architecture) over the internet, so that multiple clients can consume web services through the Web, irrespective of the type of applications or platforms.

Overview

The Web Service command is used to access and exchange information over the internet. By using this command, you can:

- Consume reusable application components as services. Examples are currency conversion, weather reports, and language translation.
- Connect to different existing applications and different platforms, irrespective of any underlying infrastructure requirements.

The Automation Anywhere Web Service establishes complete inter-polarity between clients/applications and the Web, supporting XML-based open standards, such as WSDL (Web Services Description Language), SOAP (Simple Object Access Protocol), and UDDI (Universal Description Discovery and Integration).

Using the Web Service Command

Follow these steps to create a new Web service:

1. Open the task in the Task Editor.
2. Double-click on or drag the Web Service command to the Task Actions List.

Web Service ?

Step 1: Enter complete URI

`et/country.asmx/GetCountryByCurrencyCode?CurrencyCode=` | Build URI

e.g. http://www.domainname.com/Service.asmx/Operation1?
param1=value1 & param2=value2

☑ Authentication details (For secure web services only)

Username : `JohnSmith`

Password : `************`

Domain Name :

Client Certificate : | Browse

☑ Enter Header Parameter(s)

Name : | Value : | Add

Name	Value
CountryCode	$CountryCode$

Update
Delete

Step 2: Click to test the output data | Test Output

Step 3: Provide output details

Save as XML (Optional) : `D:\John.Smith\My Documents\webse` | Browse

◉ Whole Response   ○ Selected Response

Assign the returned value
to an existing variable : `Prompt-Assignment` ▾

💡 TIP : Press F2 to insert variable.

Save Cancel

(Image as seen in Enterprise edition version 8.1)

3. **Enter complete URI:** Enter a complete URI (Uniform Resource Identifier) that you want to use for the Web Service.

4. **Authentication Details:** If you need to use secure Web Services, enter the authentication details, including user name, password, domain name, and client certificate.

5. **Enter Header Parameter(s):** *To automate a Web Service using 2 stage authentication, specify the http header parameter. Assign a variable to the parameter to support dynamic correlation of different web services. i.e. allowing the token assigned to a variable from header of one service to be passed to the header of another. Also, update or delete existing parameters.
   * Available from version 8.1 onward

6. **Build Web Service URI:** Optionally, to build a customized WSDL URI, click the Build URI button. In the Build Web Service URI window, specify input for services, operations, and parameters.

   - **Select Service, Operations and Parameter Details:** Once the connection is established, Services and Operations fields will be auto-populated. If the selected Operation has parameters, the parameters will be populated in the table with its Name, Type and Value.

     You can Add, Update or Delete the parameters for a particular operation.

(Image as seen in Enterprise edition version 8.1)

- **Raw Data Parameter(s):***Enable this option to specify the input in XML format.
  * Available from version 8.1 onward

(Image as seen in Enterprise edition version 8.1)

7. **Test Output:** Test the output data by clicking the Test Output button. You can test the output data from the specified URI.

(Image as seen in Enterprise edition version 8.1)

8. **Provide Output details:** Specify the file to which you want to save the output data.
   The output can be extracted as a whole output or as a partially extracted value. To aid this below two options are provided.

   a. **Whole Response:** When this options is selected the whole response is captured and can be stored in the selected variable.

   b. **Selected Response:** When this option is selected, only the partial response can be captured from the web service in the form of XPath and this value can then be stored to a variable.

      - **Select/View Response:** To open an XML Viewer click on this link. Select the appropriate node from the response body and the XML Path will be automatically created. Store this output into a variable, which can now be passed into the header parameter of another request. This is particularly useful for correlating two Web Services.

Also See...

- Web Service Command - Examples
- Using Advanced Settings
- FTP-SFTP Command
- Internet Connection Command
- Launch Website Command
- Send Email Command
- Web Recorder Command
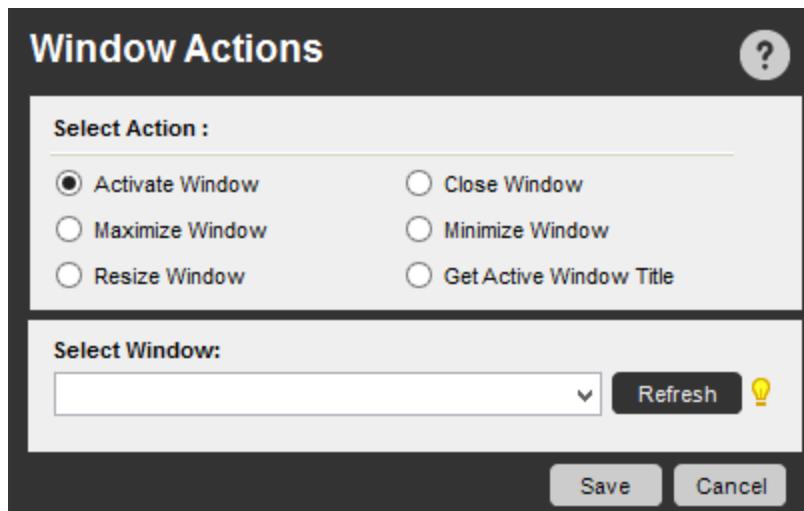
Windows Actions Command

Use this command to automate a Windows-based action, such as minimizing, maximizing, or closing an application window.

Activate Window

Use this command when you need to activate an application that is already running. Specify the window to be activated from the 'Select Window' drop-down.

> 💡Tip: Click Refresh to invoke the application you require if unavailable, yet running.

Close Window

Use this command to close an active window when running a task.

Maximize Window

Use this command to maximize an active window when running a task.

Minimize Window

Use this command to minimize an active window when running a task.

Re-size Window

Use this command to re-size an active window when running a task. Click the 'Capture' button to specify the dimensions to which the window is to be re-sized.

Get Active Window Title

Use this command to assign the active window title to a variable when running a task.
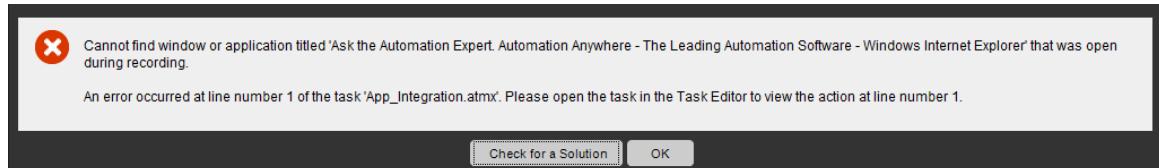
Common Errors

You may encounter the following error while using the Window Actions Command > Activate Window:

```
Cannot Find Window or Application titled <window title> that was open during recording.
```


Possible Reason(s)

The Window/Application that was open when recording the task, could be closed.

The Window/Application does not exist anymore.

Possible Solution(s)

Launch/Open Window/Application and run the task.

Check whether the Window/Application exists or not in it's destination folder/path.

Also See...

- Window Actions Command - Examples
- File and Folder Commands
- Log-to-File Command
- Manage Windows Controls Command
- Open Program/File Command
- System Command

XML Command

Feature also available in Standard edition version 8.1.2 or higher.

Use this command to process XML information that is generated from web services and cloud computing applications.

Overview

The XML command supports sessions, node editing, and Xpath expression execution, based on a tree structure of an XML document. The command enables the automated task to navigate the tree and make selections based on various criteria.

The XML command enables you to capture data that has XML formatting and saves it to a specified location.

Sub-Commands

The XML command provides the following sub-commands:

- Start XML Session
- End XML Session
- Insert Node
- Delete Node/Attribute
- Update Nodes
- Validate XML Document
- Get Node(s)
- Save Session Data
- Execute XPath Function
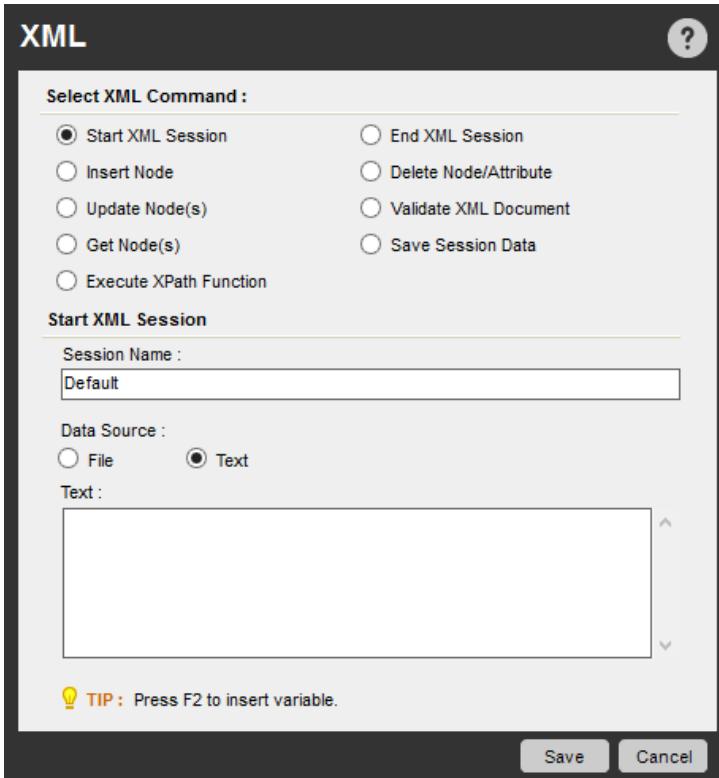
Using the Sub-Commands

To insert an XML command in your task, follow these steps:

1. Double-click or drag one of the **XML** commands to the Task Actions List pane. The XML window is displayed.
2. Specify the required parameters in the fields.
3. Click Save to save the command to your automation task.

Start XML Session Command

Use this command to specify the session name and data source (a file or text).

End XML Session Command

Use this command to complement the **Start XML Session** command and close an open XML session.

Insert Node Command

Use this command to insert a node with attributes and name spaces you specify. The location of the node is based on the position of the XPath expression.

Delete Node/Attribute Command
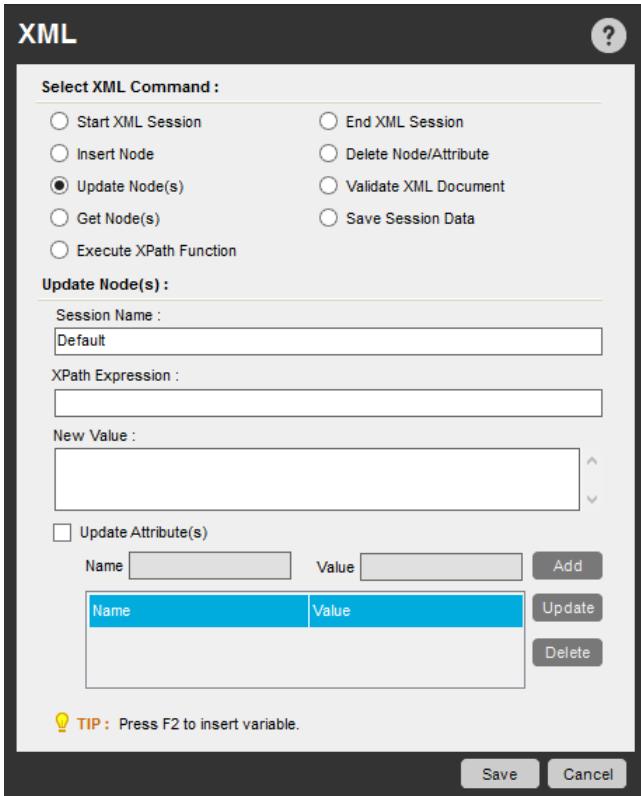
Use this command to delete a node or attribute from the XML file, as a result of specifying the XPath value.

Update Nodes Command

Use this command to update nodes in a session at the position that is specified for the XPath expression.

Validate XML Document Command

Use this command to validate session data using XML schema files (.xsd) or internal Document Type Definitions (DTDs). You can also validate whether the session data is well-formed.

You can store the output of the validation step (VALID/INVALID) in a variable. If an error occurs during validation, it is stored in the system variables named: *$Error Line Number$* and *$Error Description$*.

Get Node(s) Command

Use these commands to retrieve the values of nodes or attributes in the session data.

- **Get Single Node:** Retrieves the value of a single node or attribute from the session data, at the position specified in the XPath expression. The value is assigned to a variable.
- **Get Multiple Nodes:** Retrieves values from multiple nodes in the session data, using Text value/XPath expression/Specified attribute name, based on the specified XPath expression.

The value is assigned to a system variable named *$XML Data Node (Node name)$*, which you can use in conjunction with a LOOP command. For example, you can insert a loop to search each node in an XML data set.

Save Session Data Command

Use this command to save the session data to a variable. You can also save it to a location you specify.
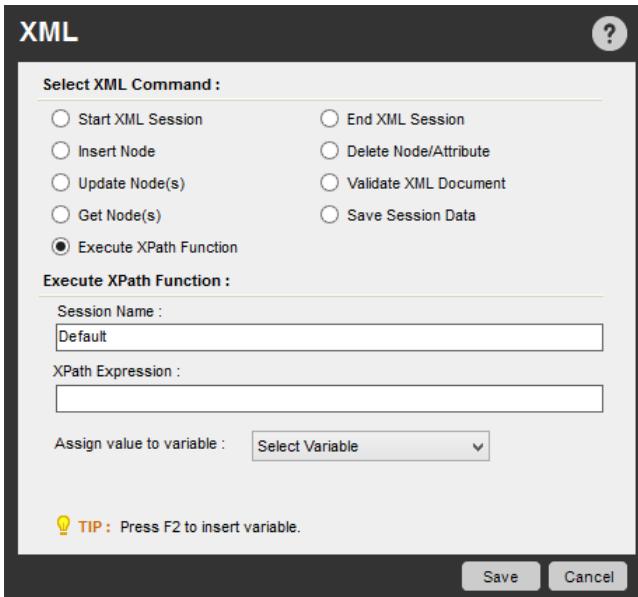
Execute XPath Function Command

Use this command to execute an XPath function and store the results in a variable.

Also See...

- XML Command - Examples
- App Integration Command
- Database Command
- Email Automation Command
- Excel Command
- Loop Command
- OCR Command
- PDF Integration Command
- Read from CSV-Text Command
- Terminal Emulator Command

Active Directory Command - Examples

Available in Enterprise edition only.

Use Case Scenario

Create a new Organizational Unit and User. Add the User to a Group in Active Directory

1. Open the Task Editor.

2. Double-click/drag and drop the 'Create User' command under the Active Directory tree from the commands list.

Active Directory

Select Command:

- ⦿ Create User
- ○ Create Group
- ○ Create Object
- ○ Search
- ○ Modify User
- ○ Modify Group
- ○ Modify Object
- ○ Get Property

Enter LDAP Path:

Parent Path: `LDAP://192.168.2.61/DC=TASERVER, DC=COM`   [...]

e.g. LDAP://DomainServer/CN=Users;DC=Domain,DC=Com

Login User: `administrator`   Login Password: `*****`

Enter New User Details:

User Name:	Password:
First Name:	Last Name: (Optional)
Display Name: (Optional)	Initials: (Optional)
E-Mail: (Optional)	Description: (Optional)
Department: (Optional)	Title: (Optional)

- ☑ User must change password at next logon
- ☐ User cannot change password
- ☑ Is Active
- ☐ Password never expires

💡 **TIP :** Press F2 to insert variable.

[Save] [Cancel]

3. Enter details in the LDAP Path section by clicking the browse button:

[...]

📝 Note: Ensure that the AD connection details are provided correctly for establishing a valid LDAP path.
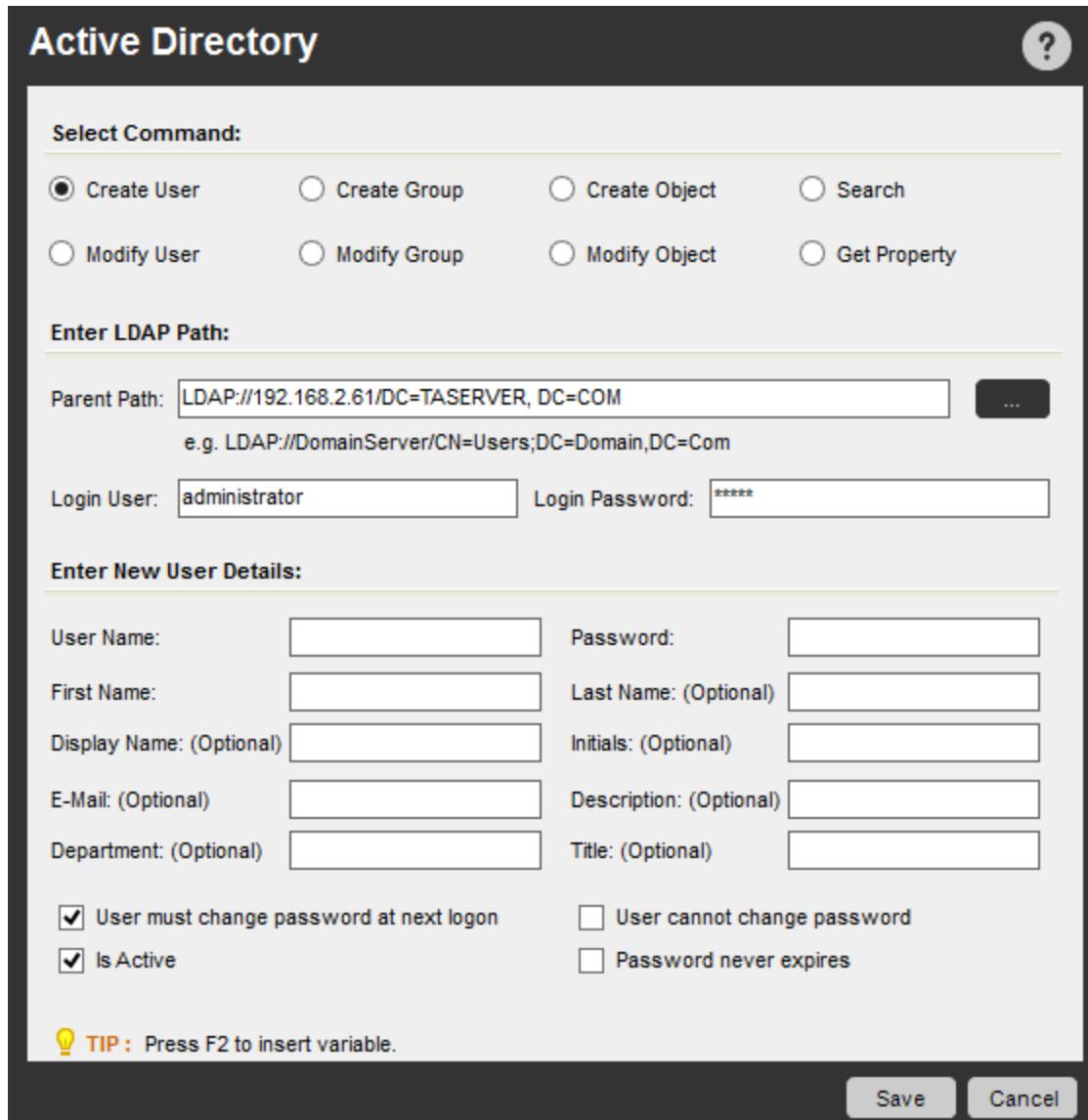
4. Follow the steps for the example.

📝 Note: Comments explain the command steps for Active Directory.

Ensure that the Organizational Unit of Sales Department does not exist in the Active Directory structure.

- **Comment:** Add a New Organizational Unit under TESTING Domain. i.e. 'SalesOU'

a. **Active Directory:** Create Object 'SalesOU' of type 'Organizational Unit'at"LDAP://sharepoint/DC=TESTING,DC=COM"

- **Comment:** Rename the newly added Object. i.e. Change the name from 'SalesOU' to 'Sales Department'

b. **Active Directory:** Rename Object 'SalesOU' To 'Sales Department' at "LDAP://sharepoint/DC=TESTING,DC=COM"

Active Directory ?

Select Command:

○ Create User ○ Create Group ○ Create Object ○ Search

○ Modify User ○ Modify Group ● Modify Object ○ Get Property

Enter LDAP Path:

Parent Path: `LDAP://192.168.2.61/DC=TASERVER, DC=COM` `...`

e.g. LDAP://DomainServer/CN=Users;DC=Domain,DC=Com

Login User: `administrator` Login Password: `*****`

Select Object Action:

Select Action: `Rename Object` ⌄

Enter Modify Object Details:

Object Name: [ ]

New Name: [ ]

Object Type: ● Computer ○ Organizational Unit

💡 **TIP :** Press F2 to insert variable.

Save   Cancel

● **Comment:** Add a Group under Sales OU. i.e. Tier1 Customer

c. **Active Directory:** Create Group 'Tier1 Customer' at "LDAP://sharepoint/OU= SalesDepartment,DC=TESTING,DC=COM"

- **Comment:** Add a new User into the Group. i.e. John Smith with logon name as 'S001'

d. **ActiveDirectory:** Create User 'S001' at "LDAP://sharepoint/OU= SalesDepartment,DC=TESTING,DC=COM"

- **Comment:** Add the newly created User (S001/John Smith) under Group named 'Tier1 Customer'

e. **Active Directory:** Add User(s) to Group 'Tier1 Customer' at "LDAP://sharepoint/OU= SalesDe-partment,DC=TESTING,DC=COM"

💡Tip: You can use the 'Disable' option for other commands, if you wish to test a specific command.

Also See...

- [Active Directory Command](#)

App Integration Command - Examples
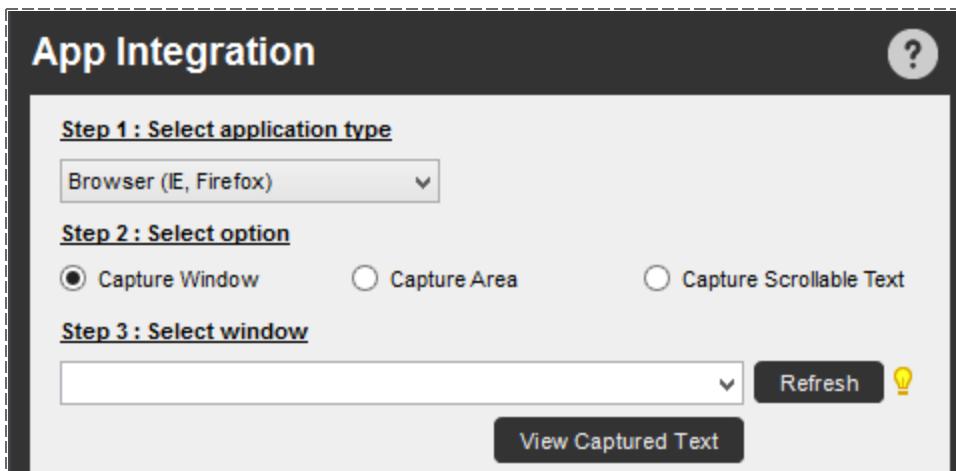
Available in Enterprise and Small Business editions only.

Capture the text from any control in an application and set the status of a check box depending on the captured text.
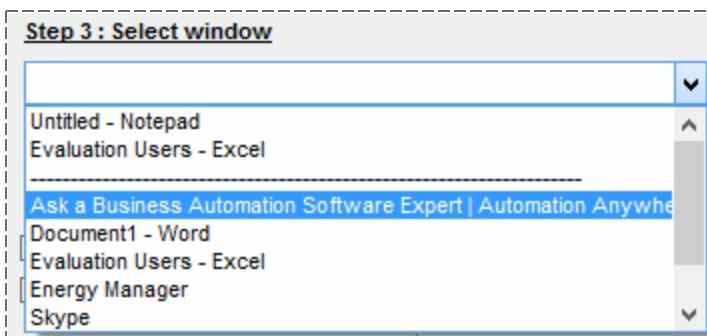
Use - Case Scenario

For instance, you want to select the check box 'Send me information on product updates' on the 'Ask the Experts' page of the Automation Anywhere website, you can do the following:
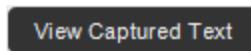
1. Open the application (in this case Browser) from which you want to capture text data.
2. Open the Task Editor in Automation Anywhere Client.
3. Double-click the App Integration command.
4. Drag and drop the Window Application option
*   The App Integration window is invoked.

5. Select the application type and options through steps 1 and 2:
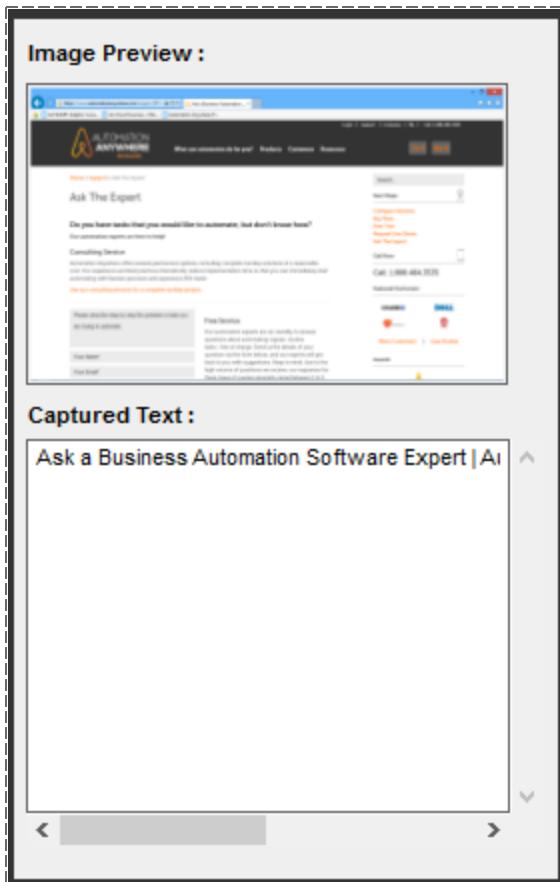
6. Select the required application window from the drop-down list.
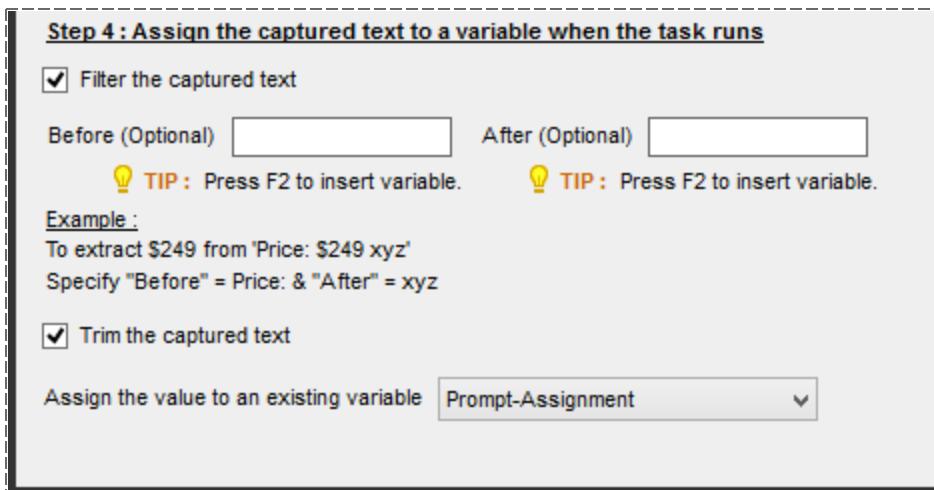
7. Click View Captured Text. This will generate the image preview on the right side, together with the captured text.

- The program provides a preview of the captured image and text (in this case the window title) in an extended panel:

8. If you need to filter the captured text, check the 'Filter the captured text' check-box and/or optionally specify 'Before' and 'After' keywords in step 4.
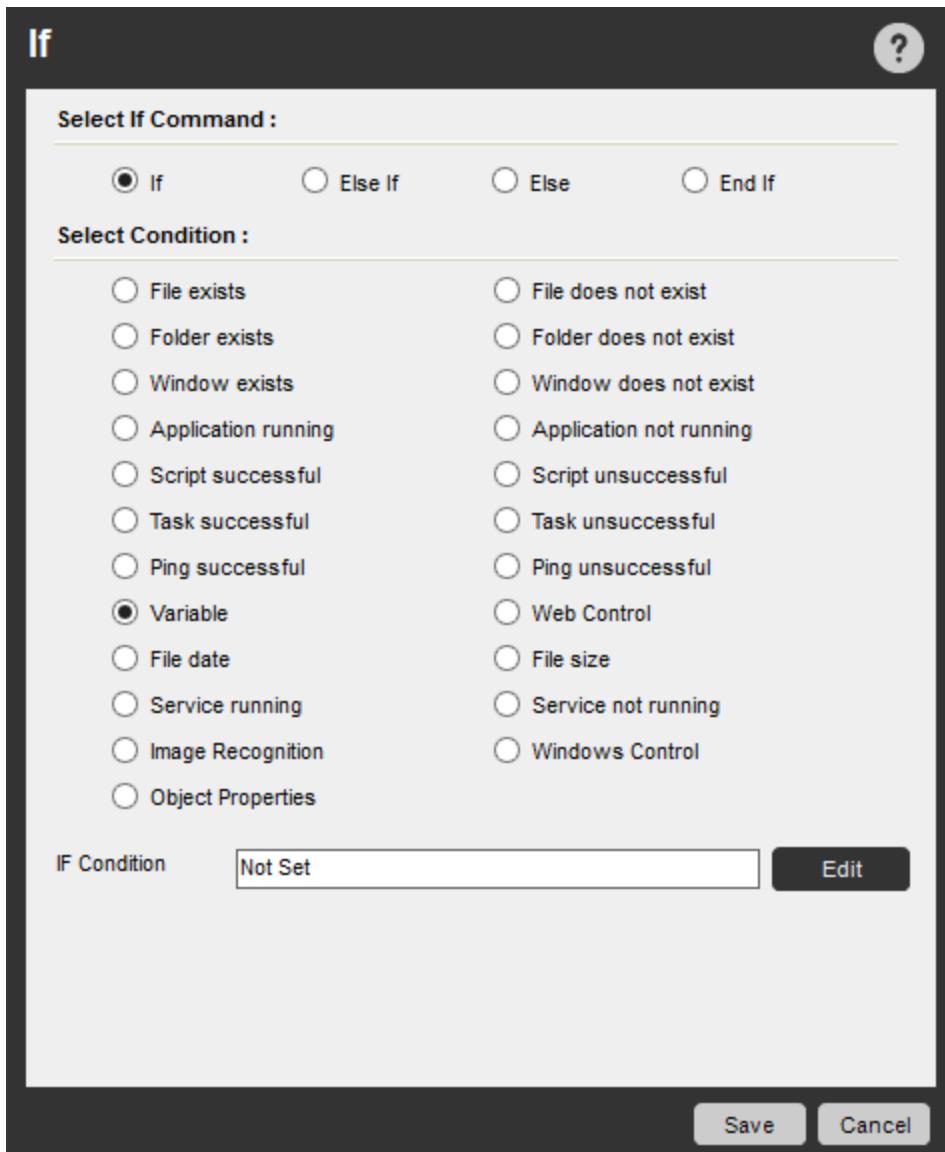
9. Assign the value to a variable from the list.
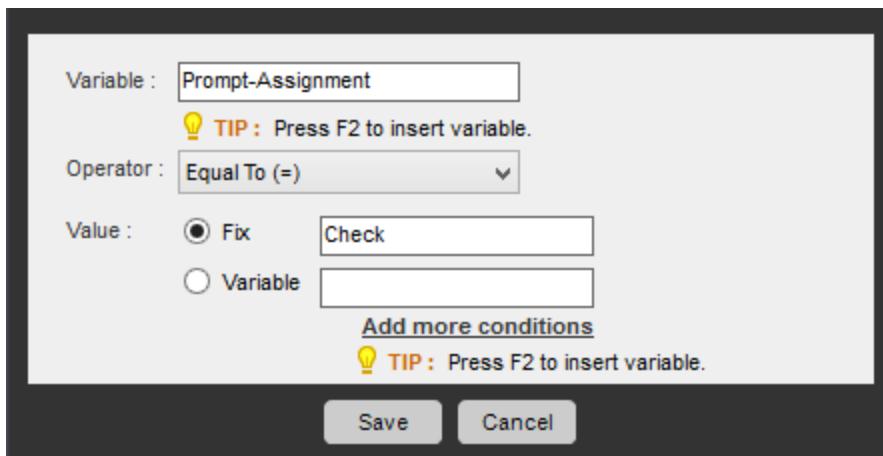
10. Click Save.

> 📝 Note: The next step involves assigning a value to the variable and then setting the status of the check - box to '√ (Checked)'. For this to work, you need to use the 'If-Else' and 'Manage Windows Controls' commands.

1. Double-click the If/Else command, and drag-drop the option 'Variable'.

If ?

Select If Command :

◉ If   ○ Else If   ○ Else   ○ End If

Select Condition :

○ File exists   ○ File does not exist
○ Folder exists   ○ Folder does not exist
○ Window exists   ○ Window does not exist
○ Application running   ○ Application not running
○ Script successful   ○ Script unsuccessful
○ Task successful   ○ Task unsuccessful
○ Ping successful   ○ Ping unsuccessful
◉ Variable   ○ Web Control
○ File date   ○ File size
○ Service running   ○ Service not running
○ Image Recognition   ○ Windows Control
○ Object Properties

IF Condition   Not Set   Edit

Save   Cancel

2. Click Edit and select the variable you specified in Step 6.

3. Select Equal To(=), select Fix, and specify the value you want to check as shown:

Variable :   Prompt-Assignment
💡 TIP : Press F2 to insert variable.
Operator :   Equal To (=)   ▾
Value :   ◉ Fix   Check
○ Variable
Add more conditions
💡 TIP : Press F2 to insert variable.
Save   Cancel

4. Double-click the Manage Windows Controls command, and double-click Check Box.

5. Select the application window from the drop-down list. Click Capture and hold and drag the mouse to the control you wish to capture (here it is 'Control Box'). The 'Manage Windows Controls' windows displays the details as follows:

6. Set the 'Check Box Status' and Click Save.

7. Save the task.

📝 Note: Your task should resemble the following image:

Using Wild Cards

You can specify wild cards (*) in a window title.

- For example, use this command to perform an operation in any window that has the word "Untitled" anywhere in its title.

- To tell Automation Anywhere Client to perform the operation in a window with any text string before or after the word "Untitled" in the window title, simply specify *Untitled* for the window title.

- Automation Anywhere Client will first search for the exact window ("Untitled"), and if it doesn't find it, it will look for windows with the word "Untitled" anywhere in the title.

- You can put the wild card character (*) at the beginning or end of the window title.

Also See...

- App Integration Command
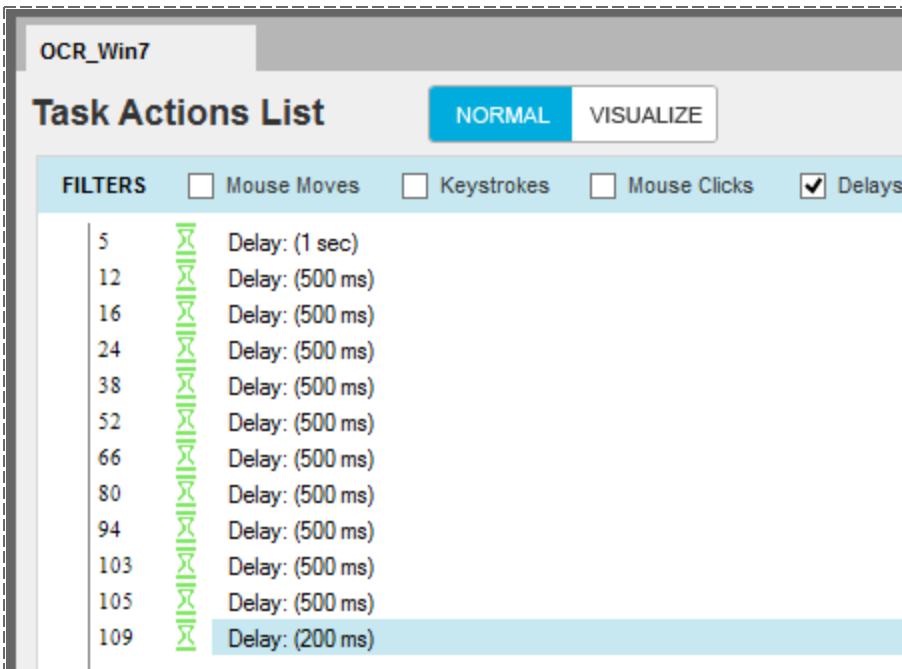
Bulk Edit Command - Examples

Use Case Scenario 1

Set a common delay of 5 seconds to multiple delay actions having varying delay criteria.
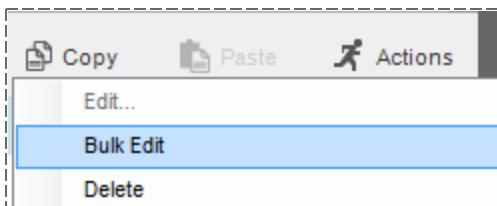
1. Open the Task Editor.
2. Select multiple 'Delay' actions or use the Filters to view all Delay actions in the Task.

3. Select all 'Delay' commands
4. Select 'Bulk Edit' from Actions drop-down or right click and select 'Bulk Edit'.

- The Delay/Wait Command window is invoked.
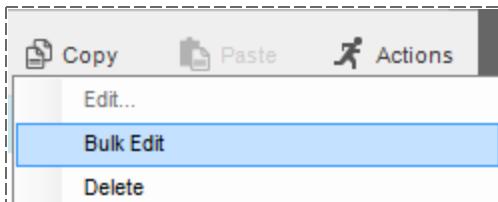
5. Type 5 in the Delay text box.
6. Select Delay in Seconds.
7. Click Save.

Use Case Scenario 2
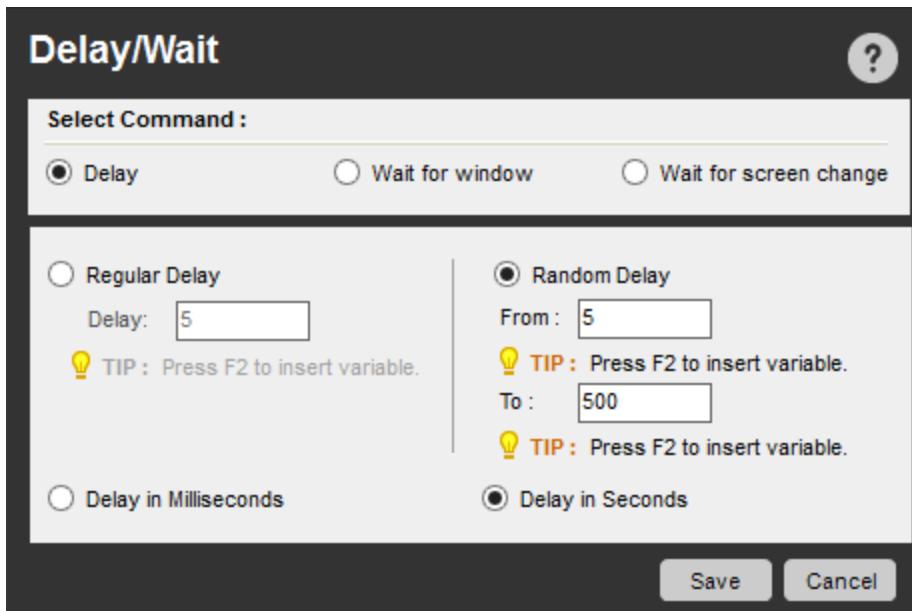
Set a common random delay to multiple 'Delay' actions having varying delay criteria.

1. Open the Task Editor.
2. Select multiple 'Delay' actions in the task.
3. Select Bulk Edit from Actions.

- The Delay/Wait Command window is invoked.
4. Select Random Delay.
5. Type 5 in the 'From' text box.

6. Type 500 in the 'To' text box.
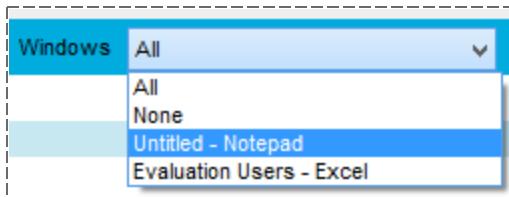
7. Select Delay in Seconds.
8. Click Save.
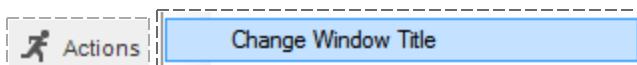
Change Window Title Command - Examples

Set a common Window title to multiple Window actions.
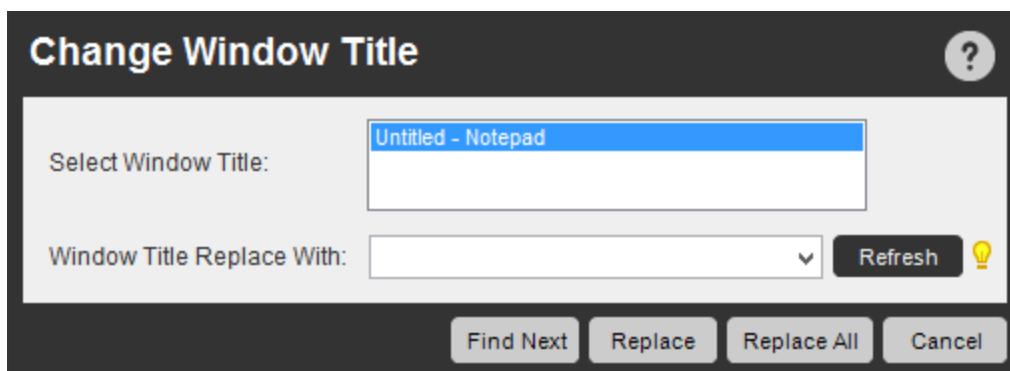
1. Open the Task Editor.

2. Select from drop down of Windows filter to select a particular Window.

3. Select the specific commands which contain windows to edit their Window Titles.

4. Select 'Change Window Title' from Actions.

5. In the ensuing Window, type a Window title with which you wish to replace selected Window titles.

6. Use 'Find' and 'Replace' or 'Replace All' to browse and replace the selected Window title.

7. Click Save.

Clipboard Command - Examples

Available in Enterprise and Small Business editions only.

This command is useful when you are transferring data using Ctrl + C (copying) from one application and Ctrl + V (pasting) into another application; or when you are performing these operations within a loop. Sometimes there is empty data in an application (e.g., Excel cells), and the copied data needs to be cleared after the previous iteration.

> 💡 Tip:
>
> 1. During a data transfer task, use the Clear Clipboard command before each iteration.
>
> 2. Before the End Loop command, insert a Clear Clipboard command.

Use Case Scenario 1

Assign the value of a variable to the Clipboard so that the value can be directly pasted using Ctrl + V after the task completes.

1. Open the Task Editor.
2. Double-click the Clipboard command.
3. Double-click 'Assign To Clipboard.'
4. Select the variable from the drop-down list whose value you want to assign to the Clipboard.
5. Click Save.

At the end of the task, if you want to move some data into the Clipboard, you can use this command so that as soon as the task finishes you can simply use Ctrl + V to paste the data into the desired application (e.g., an email client).

Use Case Scenario 2

Compare two cells of an Excel document by assigning the value of the Clipboard to a variable.

1. Open the Task Editor.
2. Double-click the Clipboard command.
3. Double-click Assign From Clipboard.
4. Select the variable Cell1 from the drop-down list.
5. You can assign values from the Clipboard only to Value type variables, not to List or Random type variables.
6. Click Save.

Similarly, assign the value of the Clipboard (after performing a Ctrl + C operation) to another variable, Cell2.

Perform the comparison operations on variables Cell1 and Cell2 as needed.

Also See...

- Clipboard Command

Clipboard Command - Examples

Available in Enterprise and Small Business editions only.

This command is useful when you are transferring data using Ctrl + C (copying) from one application and Ctrl + V (pasting) into another application; or when you are performing these operations within a loop. Sometimes there is empty data in an application (e.g., Excel cells), and the copied data needs to be cleared after the previous iteration.

> 💡 Tip:
>
> 1. During a data transfer task, use the Clear Clipboard command before each iteration.
>
> 2. Before the End Loop command, insert a Clear Clipboard command.

Use Case Scenario 1

Assign the value of a variable to the Clipboard so that the value can be directly pasted using Ctrl + V after the task completes.

1. Open the Task Editor.
2. Double-click the Clipboard command.
3. Double-click 'Assign To Clipboard.'
4. Select the variable from the drop-down list whose value you want to assign to the Clipboard.
5. Click Save.

At the end of the task, if you want to move some data into the Clipboard, you can use this command so that as soon as the task finishes you can simply use Ctrl + V to paste the data into the desired application (e.g., an email client).

Use Case Scenario 2

Compare two cells of an Excel document by assigning the value of the Clipboard to a variable.

1. Open the Task Editor.
2. Double-click the Clipboard command.
3. Double-click Assign From Clipboard.
4. Select the variable Cell1 from the drop-down list.
5. You can assign values from the Clipboard only to Value type variables, not to List or Random type variables.
6. Click Save.

Similarly, assign the value of the Clipboard (after performing a Ctrl + C operation) to another variable, Cell2.

Perform the comparison operations on variables Cell1 and Cell2 as needed.

Also See...

-   Clipboard Command

Database Command - Examples

Available in Enterprise and Small Business editions only.

Use Case Scenario 1

Connect to a database and update the records based on user input.

1. Open the Task Editor.
2. Double-click the Database command.
3. Select Connect.
4. Specify the database you want to connect to.
5. Test the connection before you save the command.
6. Similarly, insert a Database -> Insert/Update/Delete command with the appropriate SQL statement.
7. Insert a Database -> Disconnect command.

> 💡 Tip: The Insert/Update/Delete statement can include variables, and the values of the variables can be taken from the user with the Prompt command.

Use Case Scenario 2

Query all accounts with revenue over $1 million and print the account names to an Excel file.

1. Open the Task Editor.
2. Double-click the Database command.
3. Select Connect.
4. Specify the database you want to connect to.
5. Test the connection before you save the command.
6. Similarly, insert a Database -> Select Query command with the appropriate SQL statement to get all required accounts.
7. Insert a Database -> Disconnect command.
8. To access the result returned by the SQL Query, use the $Dataset Column$ system variable. Use 'Loop for Each row in a SQL query dataset' to loop through the rows of the SQL Query result. To access each column, use $Dataset Column(index)$; for example, to get the first column, use $Dataset Column(1)$.

Use Case Scenario 3

Connect to a database and create tables, drop tables, or alter tables.

1. Open the Task Editor.
2. Double-click the Database command.
3. Select Connect.
4. Specify the database you want to connect to.
5. Test the connection before you save the command.
6. Similarly, insert a Database -> Insert/Update/Delete command with the appropriate SQL statement to create, drop, or alter tables, etc.
7. Insert a Database -> Disconnect command.
8. Example 4: Connect to a database and call a procedure stored in the database to update records based on user input.
9. Open the Task Editor.
10. Double-click the Database command.
11. Select Connect.
12. Specify the database you want to connect to.
13. Test the connection before you save the command.

14. Similarly, insert a Database -> Run Stored Procedure command with the procedure name and parameters, if any. The procedure you specify should have already been created in the database.

15. Insert a Database -> Disconnect command.

> 💡 Tip: The procedure name and parameters can have variables, and the values of the variables can be taken from the user with the Prompt command.

Samples SQL Queries

- To query all columns from a table called Table1, use the query Select * From Table1.

- To query FirstName and LastName from a table called Names, use the query Select FirstName, LastName from Names.

- To query FirstName and LastName from a table called Names, where FirstName is John, use the query Select FirstName, LastName from Names WHERE FirstName=John.

Also See...
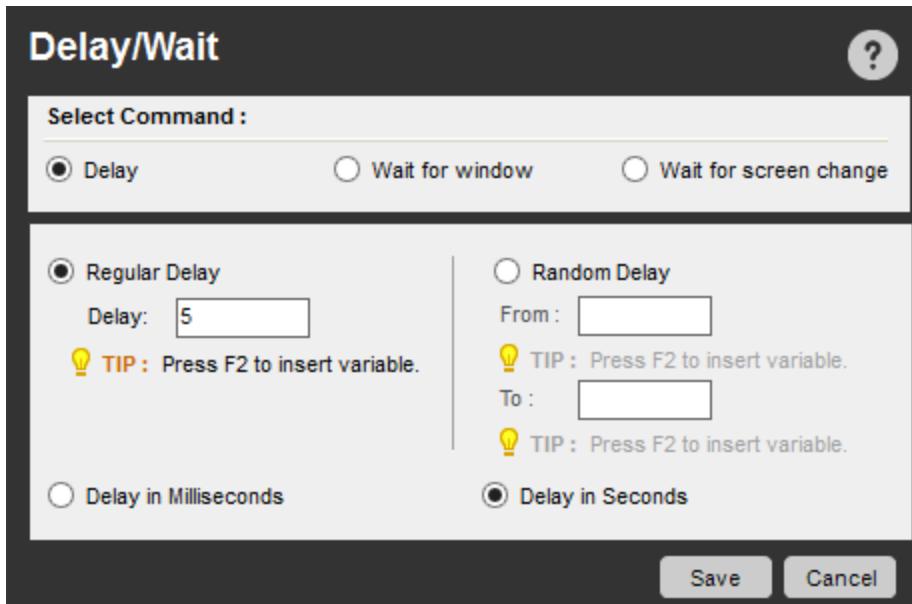
- Database Command

Delay Command - Examples

Use Case Scenario 1

Insert a delay of 5 seconds.

1. Open the Task Editor.
2. Double-click Delay.
3. Select Delay from the invoked window
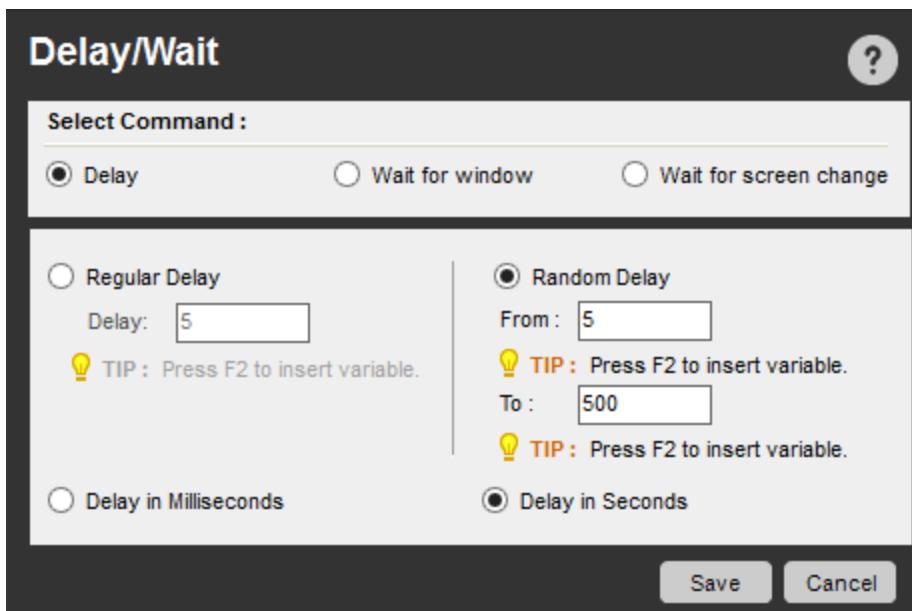
4. Type '5' in the Delay text box for the 'Regular Delay' option
5. Select 'Delay in Seconds.
6. Click Save.

Use Case Scenario 2

Generate random delays to simulate user behavior in a testing scenario.

1. Open the Task Editor.
2. Double-click the Delay command.
3. Select Delay.
4. Select the 'Random Delay' option.

5. Type 5 in the 'From' text box and '500' in the 'To' text box.

6. Select 'Delay in Seconds'.

7. Click Save.

💡 Tip: When the task runs and Automation Anywhere Client comes to the random delay command, it gen-
erates a random number in the range you specified.

Also See...

- Delay/Wait Command

Email Automation Command - Examples

Available in Enterprise and Small Business editions only.

Example 1: Get All Messages from your mail server.

1. Open the Task Editor.
2. Double-click the Email Automation command. Select the Get All Messages option.
3. Specify the Host Name. ie., your mail server ID.
4. Specify your User Name and Password.
5. Select server type. i.e., IMAP or POP3.
6. Specify the Port Number.
7. Specify the mail status if IMAP server type is selected. ie., All, Read or Unread.
8. Specify the message format. It is HTML by default.
9. Optionally, specify the folder where attachments to be saved. If not specified, attachments will not be downloaded.
10. Save the command. An Email Automation loop will be generated and you can access Email Automation variables in it.

Also See...

- Email Automation Command

Error Handling Command - Examples

Example 1: To continue extraction of all tables from a website after a table is not found.

1. Open the Task Editor.
2. Double-click the Web Recorder command.
3. Double-click the Extract Table command.
4. Type the website URL and click Launch.
5. Once the site is open, you will be redirected to the Extract Table command dialog box. Click Capture.
6. The mouse will appear as a yellow background text indicating 'Please click on the table you want to extract.'
7. Move the mouse over the table you want to extract and click anywhere in the table. (If the text background of the mouse changes to red, it means the table cannot be extracted.)
8. Specify the local file path for the CSV file where you want to store the results of the command.
9. Click Save.
10. Double-click the Loop command.
11. Double-click Times.
12. For example, if you want to extract all tables with table number between 1 and 50, specify the loop to run 50 times.
13. Click Save.
14. You need to place the Extract Table command within the above loop.
15. Double-click the Error Handling command.
16. Double-click the Begin Error Handling command.
17. Select the Continue option.
18. Select the Log Data into File check box.
19. Type the local file path for the log file, and the text you want to log.
20. Click Save.
21. The Extract Table command should be within the Begin Error Handling and End Error Handling commands.
22. When you run this command, it extracts all tables with table number from 1 to 50 in the Web page you specified, and stores the results in the CSV file you specified. If a table is not found, the error is logged in the specified log file but the task continues to extract the next table.

Also See...

- Error Handling Command

Excel Command - Examples

Example 1: Copy the values in a column in an Excel sheet to another sheet.

1. Open the Task Editor.
2. Double-click the Open Spreadsheet command.
3. Specify a session name, e.g., 'copy cells.'
4. Click [...] and select the Excel spreadsheet from which you want to copy.
5. Click Save.
6. Double-click the Open Spreadsheet command.
7. Specify a session name, e.g., 'paste cells.'
8. Click '...' and select the Excel spreadsheet where you want to paste.
9. Click Save.
10. Double-click the Get Cells command.
11. Specify a session name, e.g., 'copy cells.'
12. Select Get Multiple Cells and insert the range, e.g., if the range is from cell C6 to C18, insert C6 in From Specific Cell and C18 in To Specific Cell.
13. Choose Loop for Each row in an Excel Dataset and set the session name as 'copy cells.'
14. Double-click the Set Cells command.
15. Set the session name as 'paste cells.'
16. In the cell value, select the variable Excel Column and select the column number as 1, since you are retrieving values for one column.
17. Click Save.
18. Double-click the Go to Cells command.
19. Set the session name as 'paste cells.'
20. Select one cell below.
21. Click Save.

Example 2: To copy from Excel into Notepad.

1. Open the Task Editor.
2. Double-click the Open Spreadsheet command.
3. Click [...] and select the Excel spreadsheet that you want to copy from.
4. Click Save.
5. Click Get Cells.
6. Select Get Multiple Cells and insert the range, e.g., if your range is from cell C6 to C18, insert C6 in From Specific Cell and C18 in To Specific Cell.
7. Start a Loop for Each row in an Excel Dataset.
8. Make sure Notepad is open.
9. Double-click the Insert Keystrokes command.
10. Select the Notepad window title from the drop-down list.
11. Select the variable Excel Column from the list on the right, and insert the column number as 1, since you are retrieving values for one column.
12. Insert a special keystroke of [ENTER] after the $ExcelColumn(1)$ variable.
13. Click Save.

Example 3: Run an Excel macro in an Excel spreadsheet.

1. Open the Task Editor.
2. Double-click the Open Spreadsheet command.
3. Click [...] and select the Excel spreadsheet that contains the macro.
4. Click Save.
5. Double-click the Run Excel Macro command.
6. Type the macro name, e.g., Macro1.
7. Click Save.

Example 4: Delete an entire row in an Excel spreadsheet.

1. Open the Task Editor.
2. Double-click the Open Spreadsheet command.
3. Click [...] and select the Excel spreadsheet.
4. Click Save.
5. Double-click the Delete Cells command.
6. Specify a cell in the row you want to delete.
7. Select Entire Row from the Delete options.
8. Click Save.

NEW! Example 5: Getting values under a specified Column Title.

1. In the Task Editor double click 'Open Spreadsheet' sub-command.
2. Click [...] and select the Excel spreadsheet that hypothetically contains table with the heading titles in the first row (refer image):

	A	B	C
1	ID	Name	Designation
2	123	John Smith	Product Manager
3	124	Dwane Williams	Support Lead
4	125	Jeremy McCullum	Software Engineer

3. Click Save.
4. Select Get Cells > Get All Cells option.
5. Enable 'Contains Header' check-box to enable the task to fetch required data from the specified column.
6. Save the command.
7. Select Loop > Each Row in an Excel dataset and save. This should then populate the Task Editor with the Start Loop, Comment and End Loop lines.
8. Insert a Message Box command within this loop.
9. Press F2 and select the $Excel Column$ variable
   - A dialog box will pop-up prompting you to enter the Column Header
10. Enter Name in that text-box and click OK
    - You will notice that the Message Box Command will display $Excel Column(Name)$
11. Save the Command.
12. Save and Run the task
    - You will get the values under the Name Column one by one.

Files/Folders Command - Examples

Example 1: Copy files from one folder to another that were created within the last 180 days, and that are 50 Kb or larger.

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.
2. In the Task Editor, in the Commands list, double-click Files/Folders. A list of Files/Folders actions appears.
3. Double-click Copy Files. The Files/Folders dialog appears, with the Copy Files option selected.
4. In the Source Files text box, type the path and name of a source file to copy; or click Browse and select a file.
5. In the Destination Folder text box, type the name of a destination folder where you want to copy the file.
6. Click Advanced View. The Command Options appear on the right side of the Files/Folder dialog.
7. In the Select Size drop-down, select 'atleast', and in the KB text box type 50.
8. In the Date drop-down, select 'created date'.
9. Select 'in last' and type 180 in the 'days' text box.
10. Click Save.

Example 2: Rename all .dat files to .bak

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.
2. In the Task Editor, in the Commands list, double-click Files/Folders. A list of Files/Folders actions appears.
3. Double-click Rename Files. The Files/Folders dialog appears, with the Rename Files option selected.
4. In the File(s) text box, type the path to the folder that contains the .dat files you want to rename, and specify all .dat files by inserting the '*' wildcard character. For example: C:\MyFolder*.dat. Or click Browse and select a folder, and add '*.dat' to the folder path.
5. In the New File Name text box, type the filename extension for the renamed files; for example: *.bak.
6. Click Save.

Example 3: Back up your work at the end of the day in a Zip file.

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.
2. In the Task Editor, in the Commands list, double-click Files/Folders. A list of Files/Folders actions appears.
3. Double-click Zip Files. The Files/Folders dialog appears, with the Zip Files option selected.
4. In the Specify File(s) to Compress text box, type the path to the folder that contains the files you want to compress. For example: C:\MyWork\. Or click Browse and choose a folder.
5. Optionally, in the File Type text box, type the filename extensions for the file types you want Automation Anywhere Client to compress. Separate the filespecs with commas; for example: *.doc, *.txt, *.rtf.
6. In the Specify Destination Zip File text box, type a name for the Zip file that will hold the compressed files.
7. To choose advanced options, click Advanced View. The advanced options appear on the right side of the Files/Folder dialog. Here you can tell Automation Anywhere Client whether to overwrite or move existing files; specify a compression speed; and/or password-protect the resulting Zip file, etc.
8. Click Save.

Example 4: Delete all *.tmp files at the end of the day.

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.
2. In the Task Editor, in the Commands list, double-click Files/Folders. A list of Files/Folders actions appears.
3. Double-click Delete Files. The Files/Folders dialog appears, with the Delete Files option selected.
4. In the File(s) Name text box, type the path to the file(s) you want Automation Anywhere Client to delete. For example, to delete all temporary files with a .tmp filename extension in the C:\Temp folder, type: C:\Temp*.tmp
5. Click Save.

Example 5: Print all of the Microsoft Word documents in the folder C:\My Documents\.

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

2. In the Task Editor, in the Commands list, double-click Files/Folders. A list of Files/Folders actions appears.

3. Double-click Print Multiple Files. The Files/Folders dialog appears, with the Print Files in a Folder option selected.

4. In the Folder Name text box, type the path to the folder; for example: C:\My Documents\.

5. Optionally, in the File Type text box, type the filename extensions for the file types you want Automation Anywhere Client to compress. Separate the filespecs with commas; for example: *.doc, *.txt, *.rtf.

6. Click Save.

Example 6: Create a shortcut to open a file by clicking an icon on the desktop.

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

2. In the Task Editor, in the Commands list, double-click Files/Folders. A list of Files/Folders actions appears.

3. Select the Create File Shortcut option.

4. Specify the location of the file for which you want to create a shortcut. e.g. C:\Temp\DailyPlan.xls.

5. Specify the destination folder where you want to create the shortcut, e.g., C:\Documents and Settings\Administrator\Desktop\.

6. Click Save.

Also See...

- Files/Folders Command

FTP (File Transfer Protocol)/SFTP (Secure FTP) Command - Examples

Example 1: Download a file routinely from an FTP/SFTP server

1. Open the Task Editor.
2. Double-click the FTP/SFTP command. Select the Connect option.
3. Specify the server address, e.g., ftp://yourserver.com, and the username/password, and click Save.
4. To use the SFTP feature, check the Secure FTP box.
5. Double-click the FTP/SFTP command and select the Get File option.
6. In Remote File specify the name of the file on the FTP/SFTP server that you want to download, e.g., Folder1/Folder2/myfile.doc.
7. In Local file, select the folder name by clicking the '...' button.
8. If your document is HTML, plain text, or any other ASCII format, select ASCII format, otherwise select binary.
9. Click Save.
10. Double-click the FTP/SFTP command and select Disconnect.

Example 2: Download your Website.

1. Insert an FTP/SFTP > Connect command.
2. Insert an FTP/SFTP > Get Folder command. Specify the folder you want to download from your Web server.
3. Specify the Local Folder where you want to download the FTP/SFTP folder.
4. Optionally, specify *.htm File Type to download only .htm files.
5. Insert an FTP/SFTP > Disconnect command.

Example 3: Upload your Website.

1. Insert an FTP/SFTP > Connect command.
2. Insert an FTP/SFTP > Put Folder command. Specify the folder you want to upload to your Web server.
3. Specify the File Type that you want to upload, e.g., *.htm.
4. Insert an FTP/SFTP > Disconnect command.

Example 4: Periodically clean your FTP/SFTP folder.

1. Insert an FTP/SFTP > Connect command.
2. Insert an FTP/SFTP > Delete File(s) command.
3. Specify the File Types you want to delete, e.g., abc.htm, or *.htm, or abc*.atmx, or *.*, etc.
4. Insert an FTP/SFTP > Disconnect command.

Example 5: Rename files in your FTP/SFTP folder before you upload the new ones.

1. Insert an FTP/SFTP > Connect command.
2. Insert an FTP/SFTP > Rename File(s) command.
3. Specify the file names you want to rename, e.g., abc.htm, or *.htm, or abc*.atmx, or *.*, etc.
4. Specify the filename that you want to give to the files, e.g., OLD*.htm or OLD*.*, etc.
5. Insert an FTP/SFTP > Disconnect command.

Also See...

- FTP/SFTP Command

If-Else Command - Examples

Example 1: Close the Windows Notepad window (if open).

1. Open Windows Notepad. A new file opens automatically, titled 'Untitled Notepad.'

2. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

3. In the Task Editor, in the Commands list, double-click If/Else. A sub-menu appears.

4. Double-click Window Exists. The If dialog appears, with Window exists selected.

5. In the Select Title drop-down, select 'Untitled Notepad'.

6. Click Save.

> Note: You should insert the Close Window command between If and End if statements in the task.

Example 2: Search for a record. If found, then perform a different set of actions.

1. Automation Anywhere Client can search for an item, a word, or a record in an application, and if found, perform a set of actions that you specify; and if not found, perform a different set of actions.

2. The following example shows how to search for an item, word, or record that doesn't exist, so that you see the standard application window that says 'Not Found' (or equivalent wording).

3. Open the application, and display a 'Not Found' (or equivalent) page by searching for nonexistent data.

4. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

5. In the Task Editor, in the Commands list, double-click If/Else. A sub-menu appears.

6. Double-click Window Exists. The If dialog appears.

7. In the Select Title drop-down, select the 'Not Found' window in the application.

8. Click Save.

> Note: You would perform steps 1 to 6 to create a new condition within If and End If statements

Example 3: Make a copy of the log file named mylog.txt, if it exists.

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

2. In the Task Editor, in the Commands list, double-click If/Else. A sub-menu appears.

3. Double-click File Exists. If dialog appears with Window exists selected.

4. In the Select File text box, type (or click Browse to browse to) the path and filename of the mylog.txt file.

5. Click Save. Automation Anywhere Client inserts If File Exists and End if commands in the task.

6. In the Task Editor, in the Task Actions List, click the last action before the End If statement.

7. In the Commands list, double-click Files/Folders. A sub-menu appears.

8. Double-click Copy Files. The Files/Folders dialog appears.

9. In the Source File(s) and Destination Folder text boxes, type or browse to the path and filename of the file you want to copy, and the folder that you want to copy it to.

10. Click Save.

> Note: Automation Anywhere Client has inserted the Copy Files action immediately above the End If statement. When you run the task, Automation Anywhere Client will check if the file exists, and if it does, it will copy the file to the folder you specified.

Example 4: Create a task that prompts the user for a value, and if the user types 'STOP' the task stops.

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

2. In the Task Editor, in the Commands list, double-click Prompt. A sub-menu appears.

3. Double-click Prompt For Value. The Prompt dialog appears.

4. Click 'Assign the value to an existing variable.'

5. Click Save. Automation Anywhere Client inserts If File Exists and End if commands in the task.

> 📝 Note: This example uses variables for illustration purposes only.

6. In the Select Variable pull-down, select a variable that you previously created (for example, $User-Input$).

7. Insert an IF condition that compares the value of the $User-Input$ variable with the text 'Stop.' In the IF command, select the option 'Variable'. Click Edit. Specify the variable $User-Input$ in the first text box, select Includes, and under Fix, type 'STOP'.

8. Insert a Stop task command within the IF-End if commands.

9. Set the Repeat property of the task to 'Repeat Until I Stop.'

10. This task will continue to repeat until the user enters 'STOP' when prompted.

Example 5: To open a downloaded file, if it exists (if not, prompt for another file before proceeding).

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

2. In the Task Editor, in the Commands list, double-click Prompt. A sub-menu appears.

3. Double-click Prompt For Value. The Prompt dialog appears.

4. Select 'If-File Exists' and type the path to the file in the Select File text box, e.g., C:\Documents and Set-tings\...\Desktop\Downloadfile.txt.

5. Click Save.

6. In the Commands list, double-click Open Program/File.

7. Type the program/file path.

8. Click Save.

9. In the Commands list, double-click If-Else, then click End if in the sub-menu.

10. Select Else and click Save.

11. Double-click Prompt for File/Folder.

12. Type a custom caption and message.

13. Under Select File/Folder, select File.

14. Assign it to a list variable from the list.

15. Click Save.

16. Double-click Open Program/File again.

17. In the program/file path, press F2 and insert the list variable that was used in Prompt for File/Folder.

18. Click Save.

Example 6: Create a task that checks if a specific Web server in a cluster is up and running.

1. Open the Task Editor.

2. Double-click If-Else, then EndIf in the sub-menu.

3. Select If-Ping-Successful and type the IP address or the hostname of the first Web server in the cluster.

4. Click Save.

5. Select Else If from the If/Else menu. Then select If-Ping-Successful and type the IP address or the hostname of the next Web server in the cluster.

6. Click Save.

7. Repeat steps 5 and 6 for all Web servers in the cluster.

8. Select Else from the If/Else menu.

9. Double-click Run Task and select a task that you want to run if all Web servers are down.

10. Click Save.

NEW! Example 7: Create a task that sends notifications to Automation Anywhere trial evaluators to opt for a purchase license.

1.   In Automation Anywhere client, launch the Task Editor to create a new task.

2.   Create variables 'Trial_Period', 'Trial_Run' and 'Trial_Saves'. These shall be used in the task for If conditional commands.

3.   Now, you need to tap into customer details to find out when their evaluation package is about to expire. Use the 'Read from CSV/Text' command to achieve this.

> Note: Here we assume that a CSV file is saved to your machine that contains the required data. Also, each column in the file should contain headers/titles that match the ones specified as variables (see image)

	A	B	C	D
1	Customer Name	Trial_Period	Trial_Run	Trial_Saves
2	John Gray	14	50	50
3	Joe Black	12	50	50
4	Jane Brown	14	45	50
5	Jerry White	13	0	0

4.   Insert a 'Loop' command to read through each column and fetch the data to make the condition given in the next step true.

5.   Insert an 'If Variable' with 'Match Any' selected in the conditions window, which is invoked on clicking 'Add more conditions'.

6.   Key in the parameters such as 'Trial_Period' Equal to (=)'14' OR 'Trial_Run' Equal To (=) '50 OR 'Trial_Saves' Equal to (=) '50' for the OR condition 'Match Any'. If any one of the conditions is satisfied for a user, s/he will be notified about the pending upgrade.

Select Condition :

◉ Match Any   ○ Match All

Specify Conditions :

⊞ Add   🗑 Delete

☐ Variable	Operator	Value Type	Value / Variable
☐ Trial_Period	Equal To (=) ∨	Fixed ∨	14
☐ Trial_Run	Equal To (=) ∨	Fixed ∨	50
☐ Trial_Saves	Equal To (=) ∨	Fixed ∨	50
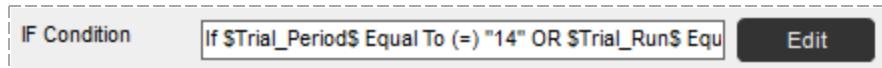
💡 TIP : Press F2 to insert variable.

Save   Cancel

- Use the 'Add' button to add as many conditions as required.

> Note: You can use only a single condition type and cannot use both i.e. Match Any + Match All.

- The 'event data' is populated in the IF command window:

IF Condition | If $Trial_Period$ Equal To (=) "14" OR $Trial_Run$ Equ | Edit

9. You will have to send notifications to your evaluation customers to opt for the purchased product version, for which you can use the 'Send Email' command.

- Insert the command between the If and End If commands.

10. Insert the End Loop at this juncture i.e. at the very end.

NEW! Example 8: Search for books by a specific author on an e-com site using the 'AND' condition

1. In Automation Anywhere client, launch the Task Editor to create a new task.

2. Create value type variables 'Books' and 'Author' to be used later into the task.

3. From the commands pane, double click/drag and drop the 'Web Recorder' command and specify url for the e-commerce site you wish to navigate to.

- You can now record the steps that lead to searching books by a specific author available on the website with the help of the 'Manage Web Controls' commands.

> 💡 Tip: Since you will be using 'Manage Web Controls' command a lot in this task, you can copy and paste where required. simply change the parameters where required!

3. Launch the e-com website with the help of 'Manage Web Control' command and in the 'Advanced View' choose 'Select Item By Text' and enter item name 'Books'. This action will enable you to select 'Books' from the drop-down list provided in the website.

4. Now you need to assign the same to a variable 'Books' which we did in the beginning. Select the 'Manage Web Controls' command and in the 'Advanced View' choose 'Get Selected Item' and 'Assign to Variable' 'Books'.

5. Next, time to key in the name of your favorite author! Use the 'Set Text' action in the 'Advanced View' of the 'Manage Web Controls' command. Key in the author name in the text box 'Text to Set'.

6. The selected author should be assigned to a variable in the next step. Use the 'Get Text' action to do this from the Manage Web Controls' command.

7. You can now use conditions to scroll through the results on different pages. Insert an 'IF variable' condition. Use the 'Match All' option in the 'If Variable' window.

8. Set parameters as shown:

- The 'event data' is populated in the IF command window:

- You can edit the parameters if you want to change those.

10. Use the 'Manage Web Controls' command to click on the 'Search' button/control. Choose 'Perform on Click' in Select action in the Advanced View.

11. You can choose to specify the number of pages to be searched on the website for which you can include a 'Loop' command. Here, let's specify '5 Times' as the loop.

12. Insert another 'Perform on Click' action here so that the task moves to the next page automatically till 5 have been reached.

> 📝 Note: Here, insert a delay of 5 or 10 seconds for the page to load completely before moving to the next one.

13. End loop.

14. Save the task.

Using Wildcards (*)

- For the options Window Exists, Window Does Not Exist, and Image Recognition, you can specify wildcards (*) in the beginning, middle and end.

- For example, if you want to perform an operation if any window has the word 'Microsoft' in the title, you can use wildcards to indicate any string before or after 'Microsoft'.

- For example, you can specify: IF Window Exists *Microsoft*.

- Automation Anywhere Client will first search for the exact window title ('Microsoft'), and if it does not find it, it will look for windows with the word Microsoft anywhere in the title.

Also See...

- If Else Command

Image Recognition Command - Examples

Feature also available in Standard edition version 8.1.2 or higher.

Example 1: Search for an image in a website and click it if found.

1. Open the Task Editor.
2. Double-click the Launch Website command.
3. Specify the URL of the website where you want to search for the image.
4. Click Launch.
5. Click Refresh.
6. Select the window where the website has been launched.
7. Click Save.
8. Assign the value to a variable from the list.
9. Double-click the Image Recognition command.
10. Click Save.

When you run the task, it will launch the website and click Image 2 if it is found within Image 1.

Using Wildcards (*)

- For the options Window Exists, Window Does Not Exist, and Image Recognition, you can specify wildcards (*) in the beginning, middle and end.
- For example, if you want to perform an operation if any window has the word 'Microsoft' in the title, you can use wildcards to indicate any string before or after 'Microsoft'.
- For example, you can specify: IF Window Exists *Microsoft*.
- Automation Anywhere Client will first search for the exact window title ('Microsoft'), and if it does not find it, it will look for windows with the word Microsoft anywhere in the title.

Also See...

- Image Recognition Command

Insert Keystrokes Command - Examples

Example 1: Insert keystrokes in a Microsoft Word document titled news.doc.

1. Open a document named news.doc in Microsoft Word.
2. In Automation Anywhere Client, click New, and in the Create New Task dialog, click Task Editor.
3. In the Commands list, double-click Insert Keystrokes or drag and drop from the command panel.
4. Click the downward-pointing arrow to the right of the Select Window text box, and in the drop-down list select news.doc - Microsoft Word.
5. Insert the required keystrokes. For example, type 'Today's tasks are complete" and under Special Keystrokes choose Enter.
6. Click Save.

Example 2: Copy a cell from an Excel document titled data.xls.

1. Open a document named data.xls in Microsoft Excel.
2. In Automation Anywhere Client, click New, and in the Create New Task dialog, click Task Editor.
3. In the Commands list, double-click Insert Keystrokes.
4. Click the downward-pointing arrow at the right of the Select Window text box, and in the drop-down list click Microsoft Excel - data.xls.
5. Follow these steps to perform a copy operation by inserting a Ctrl+C command in the Keystrokes window:
a. In the Special Keystrokes section, click Ctrl Down, then type 'c' in the Keystrokes window, and under Special Keystrokes click Ctrl Up. The Keystrokes window now displays: [CTRL DOWN]c[CTRL UP].
b. Click Save.

Example 3: Insert the current date and time.

For example, to include the date and time in filenames, so that you can identify the files that were created when you ran the task.

Automation Anywhere Client provides system variables that you can use to insert the current date and time and other useful data. The date system variables are: $Date$, $Year$, $Month$, Day, $Hour$, $Minute$, and $Second$. You can configure the $Date$ variable to print the date in your preferred format.

1. In the Task Editor, choose Tools > Variable Manager > System Variables > Date. Click the '...' button to the right of the Date variable, and in the Date format text box, edit the format, or click the down-arrow to the right of the Date format text box and select a standard format from the drop-down list.
2. For example, to name an Excel file MyExcelResults plus the current date and time, open the File Save dialog in Excel, then follow steps 1 and 2 above to insert the following in the Keystrokes text box: MyExcelResults-$Year$-$Month$-Day-$Hour$-$Minute$- $Second$.xls.
3. To configure your own date and time format, using the System variables $Year$, $Month$, Day, $Hour$, $Minute$, $Second$, follow these steps:
4. In the Task Editor, double-click Insert Keystrokes in the Commands list.
5. Click the down-arrow to the right of the Select Window text box, and from the drop-down list select the window where you want to insert the date and time.
6. From the Insert Variable list, select a system variable (e.g., Date) and click Insert. The variable appears in the Keystrokes text box.

Example 4: Insert keystrokes in any currently active window.

1. Open the window of the application where you want to enter keystrokes.
2. In the Task Editor, double-click Insert Keystrokes in the Commands list.
3. From the Select Window drop-down list, select Currently Active Window.
4. Enter keystrokes and commands in the Keystrokes text box, as described in Example 1 above.
5. Click Save. When you run the task, Automation Anywhere Client will insert your keystrokes.

Also See...

- Insert Keystrokes Command

Insert Mouse Click Command - Examples

Example 1: Perform a mouse click at the Yahoo! sign-in dialog.

1. Open mail.yahoo.com in your Web browser.

2. In Automation Anywhere Client, click New > Task Editor, or Tools > Task Editor.

3. In the Commands list, double-click the Insert Mouse Click command.

4. Click Refresh.

5. From the Select the Window... drop-down, select 'Sign in to Yahoo! - Microsoft Internet Explorer'.

6. Click Capture and hold the left mouse button. Automation Anywhere Client activates the 'Sign in to Yahoo! - Microsoft Internet Explorer' window.

7. Click the Yahoo! 'Sign In' button, and release the mouse button.

8. In the Insert Mouse Click dialog, click Save.

> Note: To insert other types of clicks (double-click, right-click, etc.), click Advanced View and
> select the appropriate options.

Also See...

- Insert Mouse Click Command

Insert Mouse Move Command - Examples

Example 1: Insert a mouse move command that moves the mouse cursor from position (200,300) to (400,500) in the active application window.

1. In Automation Anywhere Client, click New Task, Task Editor, or choose Tools > Task Editor.
2. In the Task Editor, in the Commands list, double-click Insert Mouse Move.
3. Select Mouse Position (From).
4. Click Capture. Automation Anywhere Client switches to the active application window.
5. Left-click and drag the mouse to location (200,300) and release the mouse button.
6. In the Insert Mouse Move dialog, select Mouse Position (To).
7. Click Capture. Automation Anywhere Client switches to the active application window.
8. Left-click and drag the mouse to location (400,500) and release the mouse button.
9. In the Insert Mouse Move dialog, click Save.

Also See...

- Insert Mouse Move Command

Insert Mouse Move Command - Examples

Example 1: Insert a mouse move command that moves the mouse cursor from position (200,300) to (400,500) in the active application window.

1. In Automation Anywhere Client, click New Task, Task Editor, or choose Tools > Task Editor.
2. In the Task Editor, in the Commands list, double-click Insert Mouse Move.
3. Select Mouse Position (From).
4. Click Capture. Automation Anywhere Client switches to the active application window.
5. Left-click and drag the mouse to location (200,300) and release the mouse button.
6. In the Insert Mouse Move dialog, select Mouse Position (To).
7. Click Capture. Automation Anywhere Client switches to the active application window.
8. Left-click and drag the mouse to location (400,500) and release the mouse button.
9. In the Insert Mouse Move dialog, click Save.

Also See...

- Insert Mouse Move Command

Internet Connection Command - Examples

Example 1: Connect to 'myDialUp'.

1. Open the Task Editor.

2. Double-click the Internet Connection command.

3. Select the Connect option in the 'Internet Connection' category.

4. Select your dial-up connection from the drop-down list.

5. Click Save.

Also See...

- Internet Connection Command

Launch Website Command - Examples

Example 1: Open the website www.cnn.com.

1. Open the Task Editor.

2. Double-click the Launch Website command.

3. Type 'www.cnn.com' in the 'Enter a URL and click Launch' text box.

4. Click Launch.

5. Click Refresh.

6. Select the title 'CNN.com - Microsoft Internet Explorer' in the drop-down.

7. Click Save.

Also See...

- Launch Website Command

Log to File Command - Examples

Example 1: Monitor the performance of a task by inserting a Log to File command in the task. When task playback reaches the Log to File command, Automation Anywhere Client creates a time-stamp in a log file.

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools -> Task Editor.

2. In the Task Editor, in the Commands list, double-click Log to File. The Log to File dialog appears.

3. In the Log File text box, type a path and name for the log file, or click Browse to choose an existing file.

4. Optionally, in the text box, type any text that Automation Anywhere Client should add to the log file each time it creates an entry in the file.

5. Optionally, select Add Timestamp. Each time the task runs, Automation Anywhere Client will insert the current date and time.

6. After the task runs, you can open the log file (a simple text file) and check the actions that Automation Anywhere Client has played back. Log files can help you measure the performance and accuracy of your tasks.

Example 2: Log the clipboard contents to help debug a task that copies data.

Let's say you are using loops within your tasks, and you want to check the text in the clipboard variable during each pass of the loop.

Automation Anywhere Client provides system variables called $Counter$ and $Clipboard$. You can log these variables, for example, by including a $Counter$-$Clipboard$ variables in a Log to File command. For complete information on using variables, see the Using Variables help section.

Also See...

- Log to File Command

Loop Command - Examples

Example 1: Open five Internet Explorer windows with one click.

1. Open the Task Editor.

2. Double-click the Loop command and select the option Times, then type 5 in the text box.

3. Save the command. It automatically inserts an End Loop command.

4. Insert an Open Progam/File command to open IEXPLORE.exe within the Start and End Loop commands.

Example 2: Loop through a list and insert one value at a time in an already-open Excel document.

1. Create a list variable from Tools > Variable Manager (we'll call it $my-list$).

2. Specify an option to have the variable read from a text file.

3. Let's say you have filled the values of the list variable.

4. Double-click the Loop command.

5. Select List and select $my-list$ from the drop-down list.

6. This inserts Start and End Loop commands in the editor.

7. Insert a Keystroke action $my-list$[ENTER in the Excel window. It will type the next value of the list variable each time it loops.

Example 3: Monitor an application and send email to the administrator if the application shuts down.

1. Double-click the Loop command.

2. Select Condition.

3. Select Application Running and specify the path of the application that you want to monitor for (e.g., C:\My-Application.exe).

4. Save the command. It inserts Loop and End Loop commands.

5. Insert a Send Email command after the End Loop command.

6. When you run the task, it loops as long as the application is running. As soon as the application crashes or shuts down, it leaves the loop and sends an email alert.

Example 4: Query for all contacts in the database that have last name beginning with 'R' and print the names in an Excel document.

1. Using database commands, connect to a database and issue an SQL query to fetch your records. Insert a Connect command first, then an SQL Query command.

2. Double-click the Loop command.

3. Select "Each row in a SQL Query dataset."

4. To access the result returned by the SQL query, use the $Dataset Column$ system variable provided by Automation Anywhere Client. Use the 'Loop for Each row in a SQL query dataset' option to loop through all rows of the SQL query result. To access each column, use $Dataset Column(index)$; for example, to get the first column use $Dataset Column(1)$, and so on.

5. You can use the Insert Keystrokes commands with the $Dataset Column$ variable to type the results in the Excel document.

6. Disconnect from the database.

Example 5: Print prices extracted from a pricing table on a Web page.

1. Use the Web Recorder 'Extract Table' command to extract the tabular data from the Web page.

2. After you call the Extract Table command, Automation Anywhere Client stores all of the tabular data in a system variable called $Table Column(index)$. You can use this variable to get all the rows and columns of the table you extracted.

3. Double-click the Loop command.

4. Select 'Each row in an Internet Explorer Table.'

5. To access each row and column in the $Table Column$ system variable, use the 'Loop for Each row in an Internet Explorer table' option to loop through the rows of the table. To access each column, use $Table Column(index)$; for example, to get the first column, use $Dataset Column(1)$, and so on.

6. You can use the Insert Keystrokes command with the $Table Column$ variable to type the results in the Excel document.

Example 6: Insert an Exit Loop command to break out of the loop.

1. Open the Task Editor.

2. Double-click the Loop command and select Exit Loop, and move it to the appropriate location between the Start and End Loop commands.

3. The Exit Loop command can appear only between Start Loop and End Loop statements.

Example 7: Insert a Loop for each mail message in your mail server.

1. Click Each Message in Server option.

2. Edit and fill up required details like Hostname, Username, Password, Port number and specify a folder to Save attachemnt in.

3. Save the form.

4. Edit the task. Insert variables to dowload email attributes like Subject, Message, To, Cc, Attachment.

- This loop extracts and downloads all mail details as specified.

Using Nested Loop

Let's say you are using a nested loop as follows:

1. Start Loop "List Variable my-list-variable"

2. Variable Operation: $my-list-variable$ To $Income$

3. Start Loop "5" Times

4. Keystrokes: $Income$[ENTER] in "Untitled - Notepad"

5. End Loop

6. End Loop

- You should assign the value of the list variable to a value type variable before the nested loop starts, and use the value type variable inside the nested loop.

- You can apply the same tip when you are using Loop for each row in an Internet Explorer Table, Loop for each row in a SQL Table, or Loop for each row in an Excel dataset.

Loop While Command - Examples

Example 1: Loop While for Object Property of button

1. Open the Test Editor.

2. Double-click the Loop command and select the option Times, then type 5 in the text box.

3. Save the command. It automatically inserts an End Loop command.

4. Insert an Open Progam/File command to open IEXPLORE.exe within the Start and End Loop commands.

Example 2: Loop through a list and insert one value at a time in an already-open Excel document.

1. Create a list variable from Tools > Variable Manager (we'll call it $my-list$).

2. Specify an option to have the variable read from a text file.

3. Let's say you have filled the values of the list variable.

4. Double-click the Loop command.

5. Select List and select $my-list$ from the drop-down list.

6. This inserts Start and End Loop commands in the editor.

7. Insert a Keystroke action $my-list$[ENTER] in the Excel window. It will type the next value of the list variable each time it loops.

Example 3: Monitor an application and send email to the administrator if the application shuts down.

1. Double-click the Loop command.

2. Select Condition.

3. Select Application Running and specify the path of the application that you want to monitor for (e.g., C:\My-Application.exe).

4. Save the command. It inserts Loop and End Loop commands.

5. Insert a Send Email command after the End Loop command.

6. When you run the test, it loops as long as the application is running. As soon as the application crashes or shuts down, it leaves the loop and sends an email alert.

Example 4: Query for all contacts in the database that have last name beginning with 'R' and print the names in an Excel document.

1. Using database commands, connect to a database and issue an SQL query to fetch your records. Insert a Connect command first, then an SQL Query command.

2. Double-click the Loop command.

3. Select "Each row in a SQL Query dataset."

4. To access the result returned by the SQL query, use the $Dataset Column$ system variable provided by Testing Anywhere Client. Use the 'Loop for Each row in a SQL query dataset' option to loop through all rows of the SQL query result. To access each column, use $Dataset Column(index)$; for example, to get the first column use $Dataset Column(1)$, and so on.

5. You can use the Insert Keystrokes commands with the $Dataset Column$ variable to type the results in the Excel document.

6. Disconnect from the database.

Example 5: Print prices extracted from a pricing table on a Web page.

1. Use the Web Recorder 'Extract Table' command to extract the tabular data from the Web page.

2. After you call the Extract Table command, Testing Anywhere Client stores all of the tabular data in a system variable called $Table Column(index)$. You can use this variable to get all the rows and columns of the table you extracted.

3. Double-click the Loop command.

4. Select 'Each row in an Internet Explorer Table.'

5. To access each row and column in the $Table Column$ system variable, use the 'Loop for Each row in an Internet Explorer table' option to loop through the rows of the table. To access each column, use $Table Column(index)$; for example, to get the first column, use $Dataset Column(1)$, and so on.

6. You can use the Insert Keystrokes command with the $Table Column$ variable to type the results in the Excel document.

Example 6: Insert an Exit Loop command to break out of the loop.

1. Open the Test Editor.
2. Double-click the Loop command and select Exit Loop, and move it to the appropriate location between the Start and End Loop commands.
3. The Exit Loop command can appear only between Start Loop and End Loop statements.

Example 7: Insert a Loop for each mail message in your mail server.

1. Click Each Message in Server option.
2. Edit and fill up required details like Hostname, Username, Password, Port number and specify a folder to Save attachment in.
3. Save the form.
4. Edit the test. Insert variables to download email attributes like Subject, Message, To, Cc, Attachment.
5. This loop extracts and downloads all mail details as specified.
6. Using Nested Loop
7. Let's say you are using a nested loop as follows:
8. Start Loop "List Variable my-list-variable"
9. Variable Operation: $my-list-variable$ To $Income$
10. Start Loop "5" Times
11. Keystrokes: $Income$[ENTER] in "Untitled - Notepad"
12. End Loop (for "5" times)
13. End Loop (for my-list variable)
14. You should assign the value of the list variable to a value type variable before the nested loop starts, and use the value type variable inside the nested loop.
15. You can apply the same tip when you are using Loop for each row in an Internet Explorer Table, Loop for each row in a SQL Table, or Loop for each row in an Excel dataset.

NEW! Example 8: Insert Loop While command with 'Wait' time for a 'File to Exist' when downloading data.

1. From the commands list, double click/drag and drop the 'Open File/Folder' command.
2. Insert an 'If file exists' condition.
3. Add delay of 50 milliseconds.
4. Insert 'Delete Files' command and specify the file that should be deleted. This is useful as it deletes files with same name down-loaded earlier and allows it to be saved to the same location.
5. End the 'If' command.
6. Insert a 'Web Recorder' command with 'Manage Web Controls' selected.
7. Specify the URL. The website is launched.
8. Capture the required actions - in this case, clicking the download link to save an offline copy of the data to a csv file.
9. The required parameters are populated in the 'Web Recorder' command window.
10. Insert an 'Activate Window' command to activate the website window.
11. Insert a 'Loop While' for 'File does not exists'. This ensures that the program will keep looping till the file does not exists and will loop out as soon as it finds the required file.
12. Add 'Wait time for the condition to be true'. The time set could be 5 seconds if you feel the file can be downloaded within this time frame.
13. End the loop.
14. Open the csv file and verify your data.

NEW! Example 9: Insert Loop While command with 'Wait' time for 'Ping Unsuccessful' when uploading files to a server.
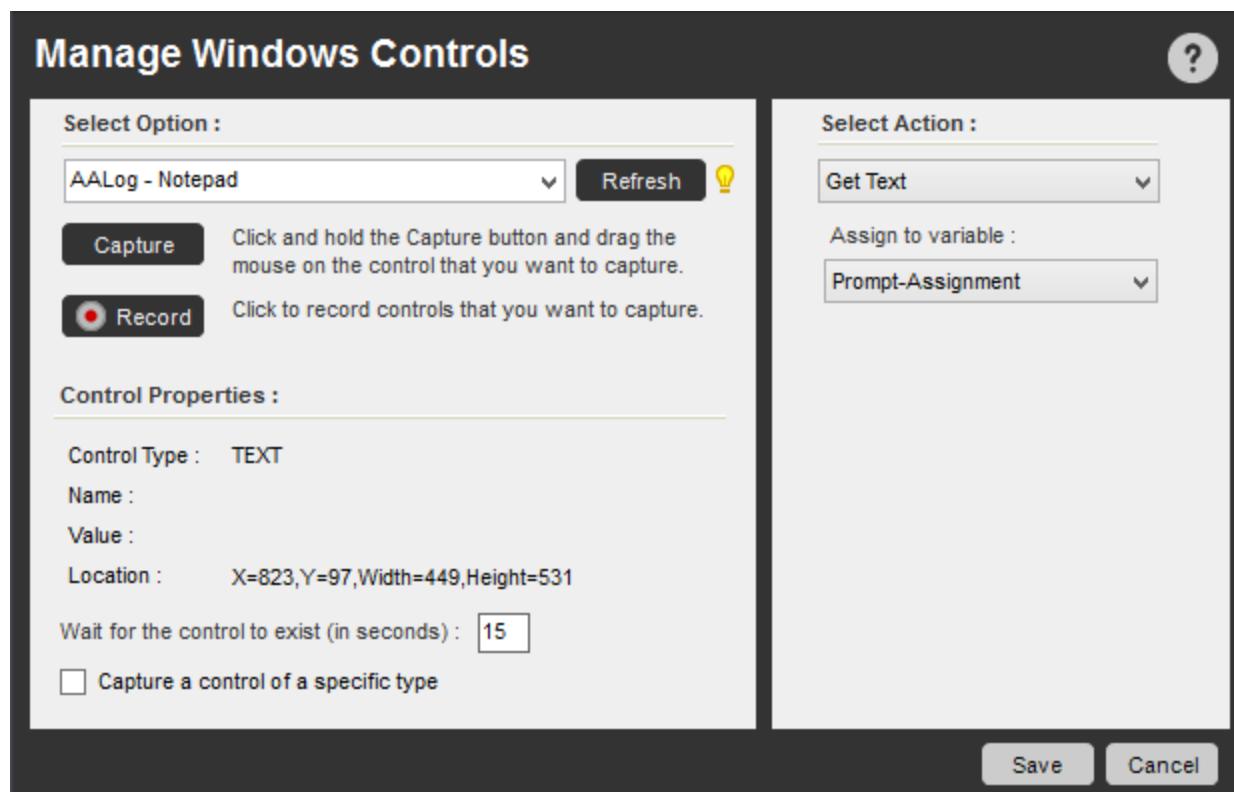
1. From the commands list drag and drop/double click the 'Loop While' command for 'Ping Unsuccessful' with 'Wait for condition to be true' enabled. This ensures that the loop will be executed till the ping is unsuccessful. The task exits the loop as soon as it validates a ping.

2. Insert 5 seconds as the time and 'End' the loop.

3. Connect to a server using the 'Connect' sub-command in 'FTP/SFTP'.

4. Use 'Start Loop' with 'Each file in Folder' option selected. Here, loop command is used to ensure that all files in the folder are selected for upload to a path specified in the next step.

5. In the 'FTP/SFTP' command select the option 'Put Files' and specify the path.

6. Transfer the data to a csv file using the 'Log to File' command.

7. End the loop.

8. Use the 'Disconnect' sub command of FTP/SFTP.

9. Now mail the data file to the user specified in the 'Send Email' command.

Manage Windows Controls Command - Examples

Available in Enterprise, Small Business and Premier editions only.

Example 1: Extract a PC's event log.
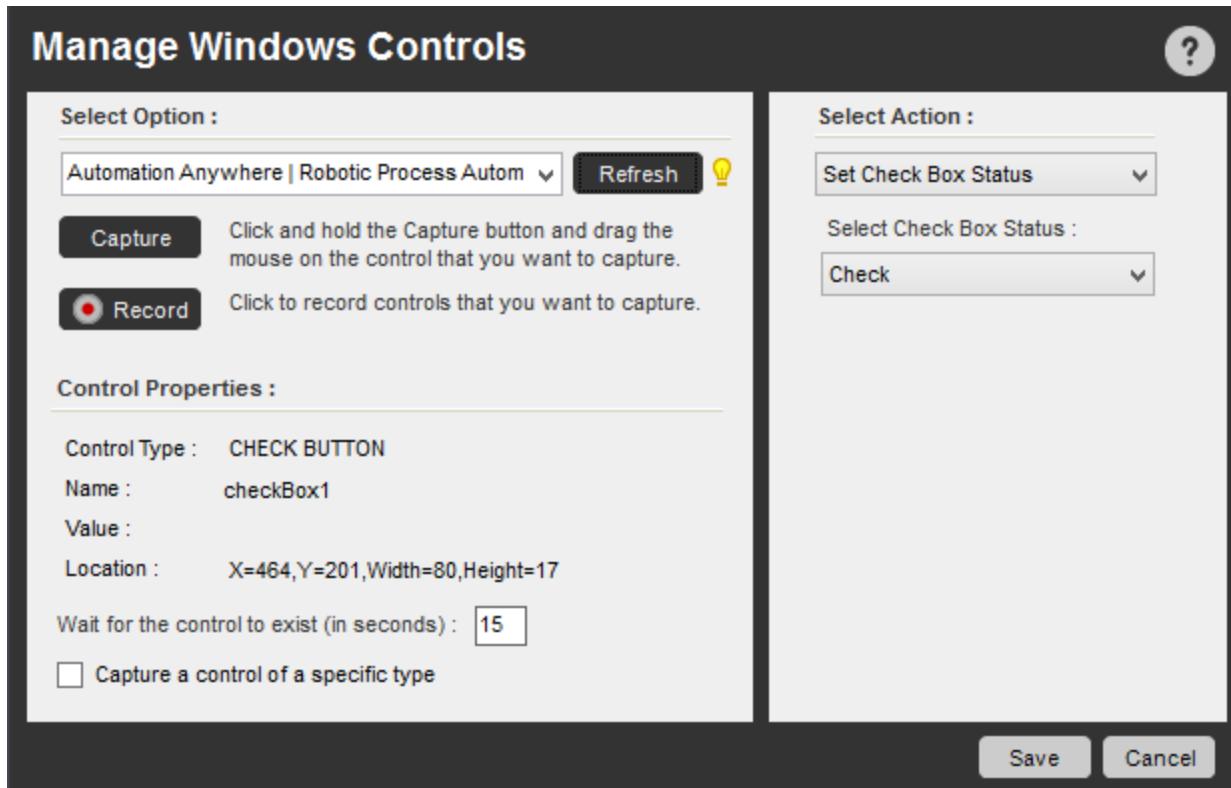
1. Open the event log whose contents you want Automation Anywhere Client to extract.

2. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

3. In the Task Editor, in the Commands list, double-click Manage Windows Controls. The Manage Windows Controls dialog appears.

4. From the Select Window drop-down list, select the window that contains the log file data you want to capture.

5. Click and hold Capture and select the text in the log file.

6. In the Manage Windows Controls dialog, select Get Text in the right pane under Select Action.

7. In the Assign to variable drop-down, select the variable to which you want to assign the log file. (To learn more about variables, see Using Variables.)

8. Click Save.

Example 2: Get (or set) the status of a check box.

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

2. In the Task Editor, in the Commands list, double-click Manage Windows Control. The Manage Windows Controls dialog appears.

3. Click and hold Capture and drag the mouse to the checkbox whose status you want to capture.

4. From the Select Window drop-down, select the title of the window that contains the checkbox whose status you want to get or set.

5. To get the status of the checkbox:

- In the Manage Windows Controls dialog, select Get Check Box Status in the right pane.

- In the Assign to variable drop-down, choose a variable to assign the checkbox status to. (For detailed information on variables, see Using Variables.)

6. To set the status of the checkbox:

- In the Manage Windows Controls dialog, select Set Check Box Status in the right pane.

- In the Select Check Box Status select Check, Uncheck or Toggle for the appropriate action you want to perform.

7. Click Save.

Example 3: Enter data from a comma-separated (CSV) data file into an application.

1. Open the application where you want to enter data.

2. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

3. In the Task Editor, in the Commands list, double-click Excel. A submenu appears.

4. Double-click the Open Spreadsheet command. The Excel dialog appears.

5. In the Excel dialog, fill in the Spreadsheet Path and Specific Sheet Name text boxes, then click Save.

6. In the Task Editor, in the Commands list, double-click Excel > Get Cells. The Excel dialog appears.

7. In the Excel dialog, select Get Multiple Cells.

8. Fill in the From Specific Cell and To Specific Cell text boxes: type the coordinates of the first and last cells of the data you want to capture.

9. Click Save.

10. In the Task Editor, double-click Loop. A submenu appears.

11. Double-click Each Row In An Excel Dataset. The Loop dialog appears.

12. Type a session name, and click Save.

13. In the Task Editor, in the Commands list, double-click Manage Windows Controls. The Manage Windows Controls dialog appears.

14. From the Select Window drop-down, select the application window where you want to insert the captured CSV data.

15. Click and hold the Capture button and drag the mouse onto the control you want to select. Then select the required action to set the value of the control to that from the Excel dataset.

16. Click Save.

17. In the Task Editor, in the Commands list, double-click Loop. A submenu appears.

18. Double-click 'Each Row In An Excel Dataset.' The Loop dialog appears.

19. Select End Loop.

20. Click Save.

21. In the commands list, double-click Excel. A submenu appears.

22. Double-click Close Spreadsheet. The Excel dialog appears.

23. Click Save.

Example 4: Select options to generate a report with an application.

1. Open the application from which you want to generate the report, and open the application's report-generation options window.

2. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

3. In the Task Editor, in the Commands list, double-click Manage Windows Controls. The Manage Windows Controls dialog appears.

4. In the Select Window drop-down, select the report generation options window.

5. Click and hold Capture and drag the mouse onto the control you want to select.

6. In the Manage Windows Controls dialog, in the right pane under Select Action, select the action to set the value of the control.

7. Click Save.

8. Repeat steps 5 to 7 for each of the options required to generate the report.

9. In the Manage Windows Controls dialog, click and hold Capture and drag the mouse in the application onto the button to be clicked to save the options you entered and begin processing the report.

10. In the Manage Windows Controls dialog, click Save.

Example 5: Extract a table from a Web page.

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

2. In the Task Editor, in the Commands list, double-click Web Recorder and double-click Open Browser. The Web Recorder dialog appears.

3. In the URL to Open text box, type the URL of the page you want Automation Anywhere Client to open.

4. Click Save.

5. In the Commands list, double-click Manage Windows Controls. The Manage Windows Controls dialog appears.

6. In Internet Explorer, open the page that contains the table you want to extract.

7. In the Manage Windows Controls dialog, in the select Window drop-down, select the Internet Explorer window.

8. Click and hold Capture and drag the mouse onto the Internet Explorer status bar.

9. In the right pane of the Manage Windows Controls dialog, in the Select Action list, select the variable to which you want to assign the value of the status bar.

10. Click Save.

11. In the Commands list, double-click If/Else. A submenu appears.

12. Double-click Variable. The If dialog appears.

13. Click Edit. The If Variable dialog appears.

14. In the Variable text box, type the name of the variable you specified above.

15. In the Operator drop-down, select Equal To (=).

16. Select Fix, and in the Fix text box, type a value.

17. Click Save.

18. In the Commands list, double-click Web Recorder. A submenu appears.

19. Double-click Extract Table. The Web Recorder dialog appears.

20. In the Step 1 text box, type the URL of the Web page where you want to capture the table.

21. In the Step 3 text box, type the name of a CSV file where you want to save the table data.

> Note: The Extract Table command above should be within the If and End If actions.

22. Click Save.

23. In the Commands list, double-click Web Recorder, and in the submenu double-click Close Browser. The Web Recorder dialog appears, with Close Browser selected.

24. Click Save.

Using Wildcards (*)

- You can use wildcards (*) when you specify a window title in any Manage Windows Controls dialog.

- For example, if you want to perform any operation in any window that contains the word Untitled,' insert the '*' wildcard character to indicate any number of characters before and/or after 'Untitled': '*Untitled*'.

- Automation Anywhere Client will first search for a window titled 'Untitled,' and if it does not find it, it will look for other windows whose names include the word 'Untitled.'

- You can insert the wildcard character (*) at the beginning or end of the window title.

Also See...

- Manage Window Controls Command

Message Box Command - Examples

Example 1: Insert Message Box commands to instruct the user how to change the screensaver on a PC.

1. Insert a Message Box command that says 'Click on Start button and then click on Control Panel.'

2. Insert a Delay of 2 seconds.

3. Insert another Message Box command that says 'Double-click on the Display.'

4. Insert another Message Box command that says 'Select the Screen Saver Tab.'

5. Insert another Message Box command that says 'Select your Screen Saver and click on Apply, then click OK.'

Example 2: Create an automated interactive task that tells the user how to perform the next action.

1. Open the Task Editor.

2. Create a task and insert a Message Box command at appropriate locations to tell the user which actions to take.

3. Insert a Pause command after the Message Box command. Sometimes users may hit OK without reading the complete message. This ensures that the task does not proceed but pauses until the user manually resumes it.

Also See...

- Message Box Command

Object Cloning Command - Examples

Available with Enterprise edition only

Use Case Scenario: To search a topic, say "Introduction to Automation Anywhere" on the Automation Anywhere support site, you can utilize the 'Object Cloning' command to perform this task.
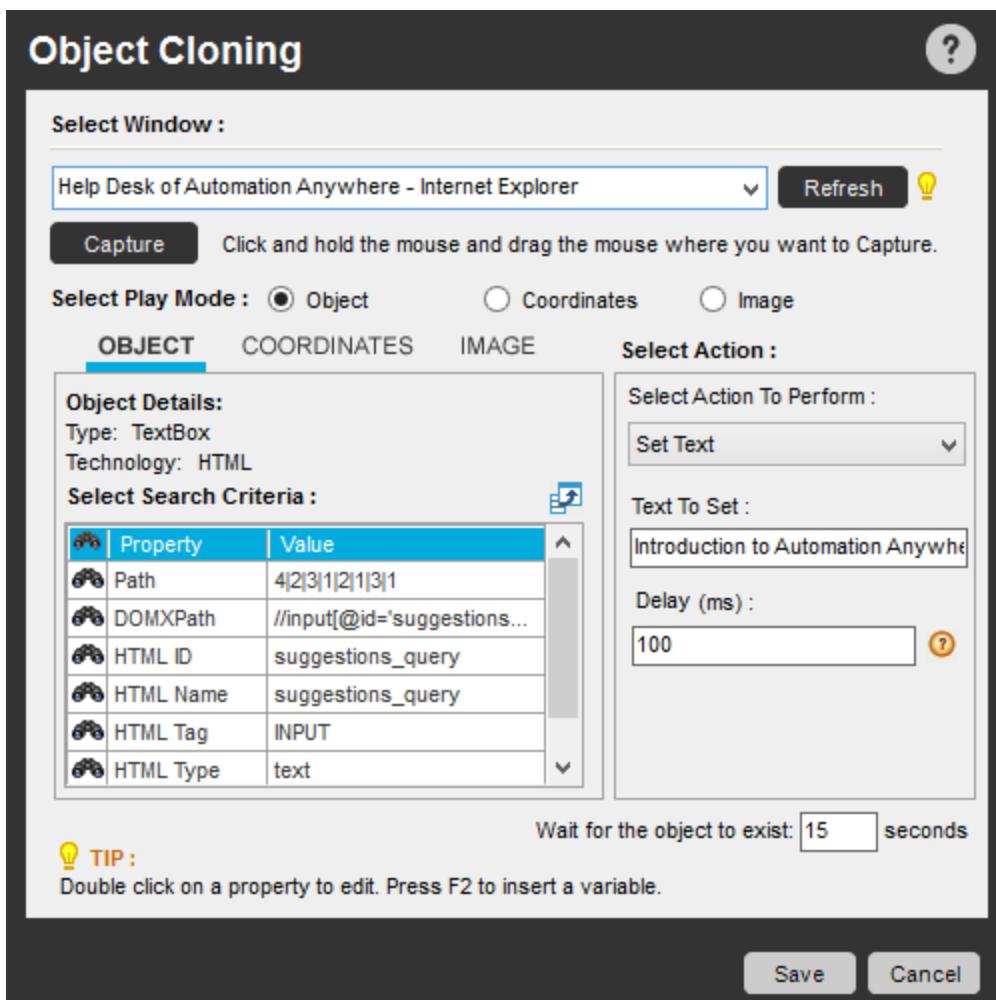
1. Click on 'New' button in the client to invoke the 'Create New Task' window.

2. Select 'Task Editor' to launch it.

3. From the Commands section, either double click or drag and drop the 'Object Cloning' command.

4. In the 'Select Window' section, from the drop down, select appropriate window; 'Help Desk of Automation Anywhere - Internet Explorer' in this case.

> Note: The window has to be open so as to be visible in the list. If open, yet unavailable, click 'Refresh'.

5. Click and hold 'Capture', and navigate the mouse to the search box in the support site that launches.
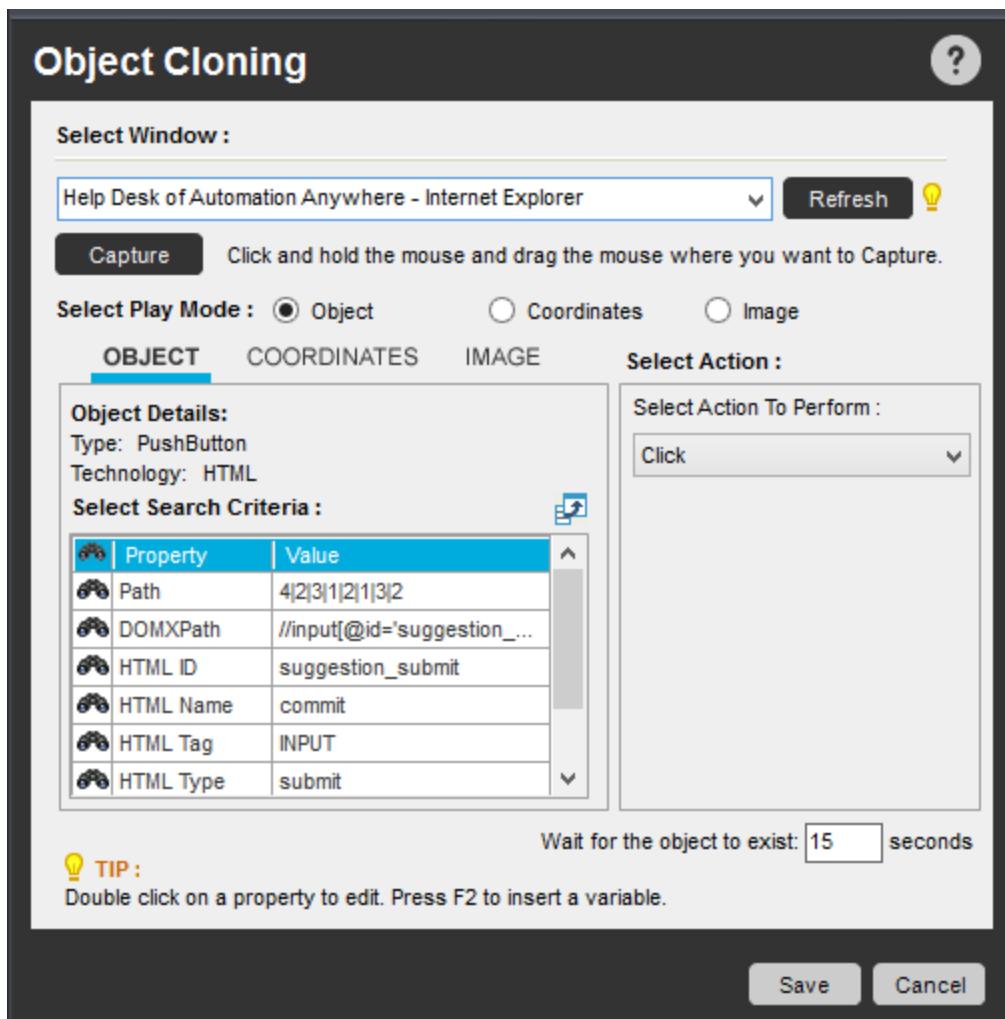
> Note: The selected object borders will be highlighted in red.

- The object properties are populated in the 'Object' tab of the 'Object Cloning' window.

6. From the 'Select Action To Perform' drop down list, choose 'Set Text' option.

7. Key in the search keywords; for instance "Introduction to Automation Anywhere" in the 'Text to Set' option.

8. Save the Command.

- The next step involves clicking the search button to navigate to appropriate pages.

9. Follow steps 3 and 4. Click and hold the capture button. Navigate your mouse towards 'Search' button.

- The 'Search' button properties are populated in the 'Object' tab.
10. In the 'Select Action' section choose 'Left Click' from the list.
11. Save the command.

Object Cloning

Select Window :

Help Desk of Automation Anywhere - Internet Explorer | Refresh

Capture   Click and hold the mouse and drag the mouse where you want to Capture.

Select Play Mode : ◉ Object   ○ Coordinates   ○ Image

OBJECT   COORDINATES   IMAGE   **Select Action :**

Object Details:
Type: PushButton
Technology: HTML
Select Search Criteria :

Property	Value
Path	4|2|3|1|2|1|3|2
DOMXPath	//input[@id='suggestion_...
HTML ID	suggestion_submit
HTML Name	commit
HTML Tag	INPUT
HTML Type	submit

Select Action To Perform :

Click

Wait for the object to exist: 15 seconds

🔆 **TIP :**
Double click on a property to edit. Press F2 to insert a variable.

Save   Cancel

- Now you need to navigate to the appropriate topic.
12. Follow Steps 3 and 4. Click and hold the capture button. Navigate the mouse to the listed topics:

Enter a Keyword to Find Resources

🔍 Introduction to Automation Anywhere   **Search**
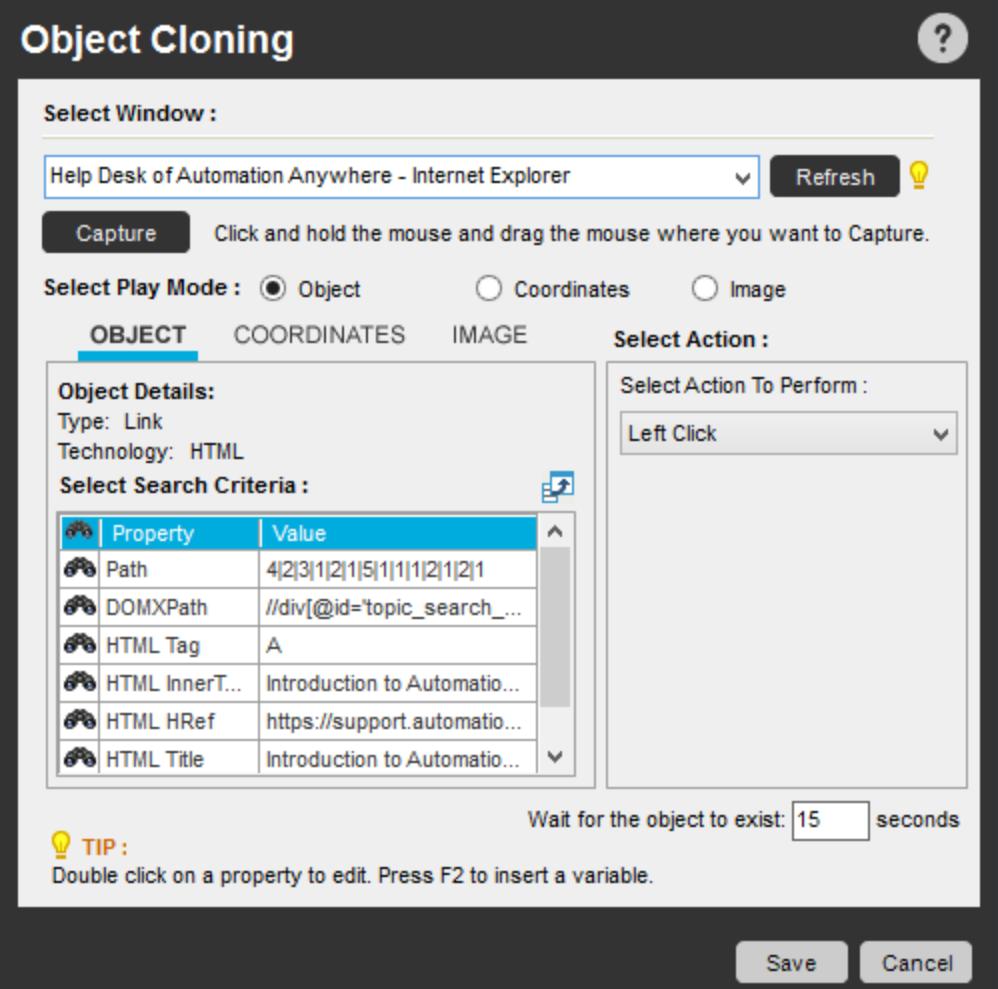
Filter your search
☑ **Documentation** ☑ **Examples** ☑ **FAQs**

📖 **Introduction to Automation Anywhere**
Introduction to Automation Anywhere Automation Anywhere allows you to create automated tasks with ease.
Development Client Documentation / Getting Started with AA »

📖 **Introduction to Process inVision**
Introduction to Automation Anywhere Process inVision Automation Anywhere Process inVision allows you to
the screen. Process inVision captures all the a...
Process inVision Documentation / Process inVision »

📖 **Getting Started**
Getting Started Getting started with Automation Anywhere (AA for short!) is easy. Using the topics in this secti
that are useful and expandable to larger, ...
Development Client Documentation / Getting Started with AA »

13. Select the topic that meets your parameters.

- The selected Object properties are populated in the 'Object' tab.

14. Go to 'Select Action' and choose 'Left Click' from the list.

15. Save the command.

16. Save the task.

You are now ready to launch your search for topics on the Automation Anywhere Support site!

Tip: Change the search parameters as and when required by editing the task.

Also See...

- Object Cloning Command

Object Recorder Command - Example

Example : To record the setup process of Launch-n-Go setup.

Open the Launch-n-Go setup file.

Click on New > Object Recorder.

Select the Launch-n-Go setup file from the Select Window option. Check the *Use Intermediate delay* box.

Click Record.

Follow the instructions of the Launch-n-Go setup by using the appropriate control property options that the Object Recorder provides during each step.

When you finish with the setup file, select Save in the Object Recorder dialog box.

Type in a name for the Task and click Save in the Save Task window.

When you run this task, it will automatically execute the Launch-n-Go setup in the background.

You can also set the SnapPoints to view your actions, step by step, using the Visualize feature.

Also See...

- Using the Object Recorder

Open Program/File Command - Examples

Example 1: Open Notepad.exe.

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

2. In the Task Editor, in the Commands list, double-click Open Program/File. The Open Program/File dialog appears.

3. In the Program/File Path text box, type 'notepad.exe'.

4. Click Save.

5. Optionally, if you want to open a file in Notepad called mynotes.txt, then type the path and filename in the Parameters text box, for example, C:\words\correspondence\mynotes.txt.

Example 2: Run a batch file from Automation Anywhere Client.

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

2. In the Task Editor, in the Commands list, double-click Open Program/File. Automation Anywhere Client opens the Open Program/File dialog.

3. In the Program/File Path text box, type the path and name of the batch file; e.g., C:\My-Batch-File.bat.

4. Click Save.

Example 3: Open a Microsoft Word document.

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

2. In the Task Editor, in the Commands list, double-click Open Program/File. Automation Anywhere Client opens the Open Program/File dialog.

3. In the Program/File Path text box, type the path and name of the batch file; e.g., C:\My Documents\smithletter.doc.

4. Click Save.

> Tip: If you use the Record feature in Automation Anywhere Client to create a task that opens an application or a file, you can make the task run faster by opening the task in the Task Editor and replacing the multiple keystrokes that you recorded with single Open Program/File commands, as described in the examples above.

Also See...

- Open Program/File Command

PDF Integration Command - Examples

Available in Enterprise edition only.

Example 1: Extract form fields from a downloaded and pre filled PDFs and split them, then merger and encrypt them.

Open the Task Editor.

See the below example with detailed steps and Comments explain the task steps.

1 Download file '$FolderPath$fw4.pdf' from 'http://www.irs.gov/pub/irs-pdf/fw4.pdf'
2 Comment: After download, fill in details of the forms. In this example two pre-filled forms are taken.
3 Comment: Below loop creates folders for the output files
4 If Folder Does Not Exist ("$OutPutFolderPath$") Then
5   Create Folder "$OutPutFolderPath$"
6   Create Folder "$OutPutFolderPath$$EmployeeData$"
7   Create Folder "$OutPutFolderPath$$DeductionSheets$"
8 End If
9 Comment: Below Loop will save images into $OutPutFolderPath$
10 Start Loop "Each File In Folder $FolderPath$"
11   PDF Integration: Convert "$FolderPath$$FileName$.pdf" to images as "$PDFFileName$_$PDFTitle$" under folder "$OutPutFolderPath$$EmployeeData$"
12   PDF Integration: Convert "$FolderPath$$FileName$.pdf" to images as "$PDFFileName$_$PDFTitle$" under folder "$OutPutFolderPath$$DeductionSheets$"
13 End Loop
14 Comment: Creation of one csv file as below to store the extract form fields.
15 Log to File: "First Name","Last Name","Home Address","City,State,Zip","Security Number","Employer","Section 10" in "$OutPutFolderPath$$CsvFilePath$"
16 Comment: Below loop will process each file from from $FolderPath$ and extract form fields to $OutPutFolder-Path$\PDFOutput.csv
17 Start Loop "Each File In Folder $FolderPath$"
18   PDF Integration: Extract Form Fields from "C:\Sample PDF\fw41.pdf"
19   Log to File: "$FirstName$","$LastName$","$HomeAd-dress$","$CityStateZip$","$SecurityNo$","$EmployerName$","$AmountForSection10$" in "$OutPutFolderPath$$CsvFilePath$"
20 End Loop
21 Comment: Below loop will process each file from $FolderPath$ and will split into two parts that is employee data and deduction sheet.
22 Start Loop "Each File In Folder $FolderPath$"
23   PDF Integration: Split "$FolderPath$$FileName$.pdf" into files as "$FileName$" under folder "$OutPutFolder-Path$\$EmployeeData$"
24   PDF Integration: Split "$FolderPath$$FileName$.pdf" into files as "$FileName$" under folder "$OutPutFolder-Path$\$DeductionSheets$"
25 End Loop
26 Comment: Merge all the documents into single file and save under $OutPutFolderPath$
27 PDF Integration: Merge Documents into "$OutPutFolderPath$\AllFormsDetail.pdf"
28 Comment: Encrypt document and save under $OutPutFolderPath$
29 PDF Integration: Encrypt Document "$OutPutFolderPath$\AllFormsDetail.pdf"; Save it as "$OutPutFolder-Path$\EncryptedAllFormsDetail.pdf"

Also See...

- PDF Integration Command

PGP Command - Examples

Available in Enterprise edition only.

Example 1: Create folders and encrypt and decrypt file using Passphrase.

1. Open the Task Editor.

2. See the below example with detailed steps and Comments explain the task steps.

3. If Folder Does Not Exist ("c:\pgp") Then
    2   Comment: Please enter the conditional commands here.
    3   Create Folder "c:\pgp"
    4 End If
    5 If File Does Not Exist ("c:\pgp\pgp.txt") Then
    6   Comment: Please enter the conditional commands here.
    7   Create File "c:\pgp\pgp.txt"
    8 End If
    9 Open File "C:\pgp\pgp.txt"
    10 Keystrokes: [SHIFT DOWN]welcome to security[SHIFT UP]command "pgp.txt - Notepad" with delay: 5 ms
    11 Keystrokes: [CTRL DOWN]s[CTRL UP] in "pgp.txt - Notepad" with delay: 5 ms
    12 Close Window: "pgp.txt - Notepad"
    13 If Folder Does Not Exist ("c:\pgp\encrypt") Then
    14   Comment: Please enter the conditional commands here.
    15   Create Folder "c:\pgp\encrypt"
    16 End If
    17 PGP: Encrypt Files using Passphrase; Source: "C:\pgp\pgp.txt"; Destination: "C:\pgp\encrypt"
    18 If Folder Does Not Exist ("c:\pgp\decrypt") Then
    19   Comment: Please enter the conditional commands here.
    20   Create Folder "C:\pgp\decrypt"
    21 End If
    22 PGP: Decrypt Files using Passphrase; Source: "C:\pgp\encrypt\"; Destination: "C:\pgp\decrypt\"

Example 2: Create folders and encrypt and decrypt file using Keys after creating the key files first.

1. Open the Task Editor.

2. See the below example with detailed steps and Comments explain the task steps.

1 If Folder Does Not Exist ("c:\pgp") Then
2   Comment: Please enter the conditional commands here.
3   Create Folder "c:\pgp"
4 End If
5 If File Does Not Exist ("c:\pgp\pgp.txt") Then
6   Comment: Please enter the conditional commands here.
7   Create File "c:\pgp\pgp.txt"
8 End If
9 Open File "C:\pgp\pgp.txt"
10 Keystrokes: [SHIFT DOWN]welcome to creation of key files [SHIFT UP]bc in "pgp.txt - Notepad" with delay: 5 ms
11 Keystrokes: [CTRL DOWN]s[CTRL UP] in "pgp.txt - Notepad" with delay: 5 ms
12 PGP: Create Keys; Public key: "C:\pgp\public.pkr"; Private key: "C:\pgp\private.skr"
13 Close Window: "pgp.txt - Notepad"
14 If Folder Does Not Exist ("c:\pgp\encrypt") Then
15   Comment: Please enter the conditional commands here.
16   Create Folder "c:\pgp\encrypt"
17 End If
18 PGP: Encrypt Files using Public Key; Source: "C:\pgp\pgp.txt"; Destination: "C:\pgp\encrypt"
19 If Folder Does Not Exist ("c:\pgp\decrypt") Then
20   Comment: Please enter the conditional commands here.
21   Create Folder "C:\pgp\decrypt"
22 End If
23 PGP: Decrypt Files using Public Key; Source: "C:\pgp\encrypt\"; Destination: "C:\pgp\decrypt\"

Also See...

- PGP Command

Play Sound Command - Examples

Example 1: Insert a Play Sound command to beep when the task finishes.

1. Open the task in the Task Editor.

2. Double-click the Play Sound command and select Play Beep option.

3. Click Save.

4. Move the Play Beep action to the last position in the task.

Example 2: Play a recorded file.

1. Open the task in the Task Editor.

2. Double-click the Play Sound command, select Play Media File option and specify a file to play.

3. Select the media file to play, and click Save.

Also See...

- Play Sound Command

Printers Command - Examples

Example 1: Change the default printer.

1. Open the Task Editor.
2. Double-click the Set Default Printer command.
3. Select a printer from the list of installed printers.
4. Click Save.

Example 2: Remove a printer.

1. Open the Task Editor.
2. Double-click the Remove Printer command.
3. Select a printer from the list of installed printers.
4. Click Run script command.
5. Click Save.

Also See...

- Printer Settings Command

Prompt Command - Examples

Example 1: Prompt the user for Start and End dates and generate a sales report for the date range.

1. Open the Task Editor.
2. Double-click the Prompt command and double-click Prompt For Value.
3. Select Don't Write In Any Window from the drop-down list.
4. Type a message to prompt the user for the Start date.
5. Assign the value the user enters to a variable, by selecting the variable from the variable list.
6. Similarly, assign the End date entered by the user to a variable.
7. You can use the two variables in place of start and end dates in the task that generates the sales report.
8. When the task runs, the two variables will be assigned the values entered by the user.

Example 2: Create a task that prompts the user for data and automates the installation of an application.

1. Click Record in Automation Anywhere Client.
2. Perform the steps to install the application.
3. If you plan to run the task on different computers with different resolutions, we suggest using the keyboard instead of the mouse while recording the installation process.
4. After you finish recording and save the task, select it in the Task List and click Edit.
5. Review your keystroke actions, such as the actions where you specified an installation folder, etc.
6. Right-click an action and select Convert to Prompt (or select it and click the Actions button).
7. This converts your recorded keystrokes to a Prompt command that prompts the user when the task runs.

In this manner, you can create a generic task to automate the application and not depend on the values you entered while recording.

Example 3: Prompt for a file that will be used in a loop to enter data.

1. Open the Task Editor.
2. Double-click the Prompt command and double-click Prompt for File.
3. Type the caption text.
4. Enter a custom message for the user to specify the file.
5. Assign the value to a list variable from the list.
6. Expand the Loop command and double-click the list.
7. Select the list variable you assigned to the file.
8. Double-click the Insert Keystrokes command.
9. Select the window title from the list of files where the data will be entered.
10. Type the data you want to insert into the file.
11. Click Save.

> Note: This is a procedure for a single file. You can do the same for multiple files; you will have to check 'Enable this task to run with similar window titles/files' under the General Tab, in the Properties menu.

Example 4: Prompt for the folder that contains a setup file (setup.exe).

1. Open the Task Editor.
2. Double click the Prompt command and double-click Prompt for Folder.
3. Type a caption.
4. Type a message for the user to select the folder.
5. Assign the value to a value variable from the list.

6. Click Save.

7. Double-click the Open Program/File command.

8. In the Program/File path, type the variable to which you assigned the value of the folder, followed by '\setup.exe'.

9. Click Save.

Example 5: Prompt the user for confirmation before deleting a record from a database.

1. Open the Task Editor.

2. Insert the command to connect to a database.

3. Double-click the Prompt command and double-click Prompt For Yes/No.

4. Type a message for the user to select the folder.

5. Type a caption.

6. Type a custom message asking the user to select the desired option (Yes/No).

7. Assign the value to a value variable from the list.

8. Click Save.

9. Double-click the Loop command and double-click Condition.

10. In the If dialog, choose File exists.

11. Insert an If - Variable condition to check whether the variable to which the value assigned has the value Yes.

12. Click Save.

13. Double-click the Database command and double-click Insert/Update/Delete.

14. In the Database dialog, in the Enter Insert/Update/Delete Statement text box, type the delete statement to delete a record from a particular table.

15. Click Save.

16. In the Program/File path, type the variable to which the value of the folder was assigned, followed by \setup.exe.
Double-click the Database command and double-click Disconnect.

17. Click Save.

Also See...

- Prompt Command

Read From CSV/Text Command - Examples

Example 1: Read values from a tab-delimited CSV file and transfer the values to another application.

1. Open the Task Editor.

2. Double-click the Read from CSV/Text command.

3. Specify the name of the CSV file you want to read.

4. Select Tab as the delimiter, and check the Header check box if your CSV file has headers.

5. Click Save.

6. To access the values returned by the command, use the $Filedata Column$ system variable. Use the 'Loop for Each row in a CSV/Text file' option to loop through the rows of the result. To access each column, use $Filedata Column(index)$; for example, to get the first column, use $Filedata Column(1)$.

7. Within the loop, insert Manage Windows Controls commands to set the value of the controls in the target application.

NEW! Example 2: Getting values under a specified Column Title.

1. In the Task Editor double click 'Read from CSV/Text' command.

2. Click [...] and select the CSV File that hypothetically contains table with the heading titles in the first row (refer image):

	A	B	C
1	ID	Name	Designation
2	123	John Smith	Product Manager
3	124	Dwane Williams	Support Lead
4	125	Jeremy McCullum	Software Engineer

3. Tick the Contains Header check-box.

4. Save the command.

5. Double click on Loop > Each Row in A CSV/Text File Command.

6. Save the command; it should then populate the Task Editor with the Start Loop, Comment and End Loop lines.

7. Inside the loop, double click the Message Box Command

8. Press F2 and select the $Filedata Column$ variable

- A dialog box will pop-up prompting you to enter the Column Header

9. Enter Name in that text-box and click OK

- You will notice that the Message Box Command will display $Filedata Column(Name)$

10. Save the Command.Save and Run the task; you will get the values under the Name Column one by one.

Registry Command - Examples

Example 1: Create a new registry key, before creating a key check out whether it is already created or not.

1. Open the Test Editor.
2. Insert Checkpoint - Registry, select "Registry key exists" option and provide a registry key to check if the registry key exists or not.
3. Double-click the Registry command.
4. Enter the registry key to be created.
5. Enter a value name of the key to be created.
6. Select a type of a value.
7. Enter appropriate data for the value according to the selected value data base.
8. Click Save.
9. Save the test.

Example 2: Get data of a value from registry

1. Open the Test Editor.
2. Double-click the Get Value Data command.
3. Specify the registry key for which data needs to be fetched.
4. Assign the value to a variable from the list.
5. Click Save.
6. Save the test.

Example 3: Export the registry

1. Open the Test Editor.
2. Double-click the Export command.
3. Select the export range.
4. Provide the path where you want to save the exported registry file.
5. Click Save.
6. Save the test.

Run Script Command - Examples

Example 1: Summation of all Variables using Java script.

1. Create and save a Java Script to sum all variables.

2. Create user defined variables using variable manager. For example:
   Variable as 0 and variable1, variable2, variable3, variable4, variable5 as NULL.

3. Now Using Variable operation command
   Assign Variable1 = Variable + 1
   Variable2 = Variable1 + 1
   Variable3 = Variable2 + 1
   Variable4 = Variable3 + 1
   Variable5 = Variable4 + 1

4. Click Run script command.

5. Specify the saved Java Script in Insert Script.

6. Insert assigned variables in parameters field. Insert a variable Return value field.

7. Drag and Drop a message box command, Using F2 support insert the variable which will return the sum.

8. Save the task.

9. Result: When you run this task, it will run the script and return a message box with summation of all variables.

Example 2: Loop through all files in a folder, and pass each filename as a parameter to a VBScript that checks the size of the file and prompts the user.

1. Double-click the Loop command.

2. Select 'Loop for each file in a folder.'

3. Specify the path of the folder from which you want to pass all the files as a parameter to the VBScript.

4. Call the VBScript within the loop and pass the following system variables as a parameter to the script: $CurrentDirectory$, $FileName$.$Extension$.

5. Save the task.

Example 3: Create Run time database to insert records into database (.mdb) file from an Excel operation or a Database task.

1. Create and save a script in VBScript to create a database (.mdb) file.

2. Edit the required Excel operation or Database task.

3. Select the task action where you wish to create a database(.mdb) file.

4. Click Run script command.

5. Specify the saved VBscript in Select Script. Click Save.

6. You can then specify Loop and Database operations to read required Excel data and insert the data into the database file (.mdb)

7. Save the task.

- When you run this task, it will run the script and create a database file(.mdb) after which data will be inserted from Excel into the database(.mdb) file.

Example 4: Add data into an XML file from an Excel operation.

1. Create and save a script in VBScript to add data into XML file.

2. Edit the required Excel operation.

3. Specify Loop and Database operations to get required Excel data.

4. Select the task action where you wish to add Excel data into XML file.

5. Click Run script command.

6. Specify the saved VBscript in Select Script.

7. Insert variables in Parameters.

8. Click Save.

9. Save the task.

- When you run this task, it will run the script and add Excel data into the specified XML file.

Also See...

- Run Script Command

Run Task Command - Examples

Example 1: Create a process (Task Chain) that calls several smaller tasks.

1. Open the Task Editor.

2. Double-click the Run Task command.

3. Click Browse, and select the path to notepad.atmx.

4. You can select repeat and speed properties for the task.

5. Click Save.

6. Insert Run Task commands to call the remaining tasks.

```
Note: You can run as many tasks as you want from a single task. You can also chain tasks, e.g.
call Task2 from Task1, and call Task3 from Task2, etc. You can even call tasks conditionally; for
example you can call the Run Task command within an IF condition.
```

Also See...

- Run Task Command

Screen Capture Command - Examples

Example 1: Capture a screen of an active application.

1. Open the application that you wish to capture.
2. Open the Task Editor in Automation Anywhere Client.
3. Double-click the Screen Capture command.
4. Choose 'Capture Window' option.
5. Select the application window from the drop-down list.
6. Provide the path where you want to save the captured image of the application screen.
7. Click on 'Overwrite File' to overwrite the output if desired.
8. Click on Save.
9. Save the task.

Also See...

- Screen Capture Command

Send Email Command - Examples

Example 1: Send an email to a list of contacts, and attach a document with details about a new product.

1. Create a List variable for your list of contacts. You can enter all of the list values in the Variable Manager, or you can read the data from a text file.
2. Open the Task Editor.
3. Insert a Loop for List variable action from the Loop command. Select the list variable that you created in step 1.
4. Double-click the Send Email command.
5. Specify the appropriate From address.
6. In the To text box, type the list variable you created in step 1.
7. Specify the path and name of the document you want to attach.
8. Type the email message you want to send.
9. You can use the following keywords to send specific information about the task: <taskname>, <desc>, <repeat>, <status>, <lastruntime>.
10. Save the command.
11. Place the Send Email command within the Start and End loop.
12. When you run the task, it loops through the list and sends an individual email to each contact, with the attached document.

Also See...

- Send Email Command

Services Command - Examples

Example 1: Start a user-defined or Windows service, if it is not already running, before running an application that requires the service to be running.

1. Open the Task Editor.
2. Double-click the Get Service Status command.
3. Select the service from the list of available installed services.
4. Select the variable from the list to assign the value to.
5. Click Save.
6. Insert an IF condition that checks whether the value of the variable selected in step 4 is not equal to Started. In the IF command, select the option Variable. Click Edit. Type the variable in the first text box, select the operator Not equal To (<>), and in the Fix option, type Started.
7. Double-click the Start Service command and insert the same actions within the If-Endif commands.
8. Select the service from the list of available installed services.
9. Click Save.
10. Double-click the Open Program/File command.
11. Specify the name of the application you want to run.
12. Click Save.

Example 2: Pause a service before performing an action, then resume it.

1. Open the Task Editor.
2. Double-click the Pause Service command.
3. Select the service from the list of available installed services.
4. Click Save.
5. Insert the commands for the actions you want to perform.
6. Double-click the Resume Service command.
7. Select the service from the list of available installed services.
8. Click Save.

Also See...

- Services Command

Simple Network Management Protocol (SNMP) Command - Examples

Example 1: Send a SNMP Get message to agent to access its computer information.

1. Open the Task Editor.
2. Double-click the SNMP Get command.
3. Specify the name or IP address of Agent ie., newpc or 132.18.4.12
4. Specify the Community that the Agent belongs to.
5. Specify Object ID.
6. Select the required Data Type format options, ie., Ascii or Hex.
7. Specify a path for the log file.
8. You can assign the result value to a variable.
9. In Advanced View, specify the SNMP version V1 or V2C.
10. Specify the Timeout required in milliseconds ie., 1000
11. Specify the Remote port.
12. Specify the number of retry times for the message.
13. Click Save.

Example 2: Send Trap message to SNMP Manager.

1. Open the Task Editor.
2. Double-click the Send Trap command.
3. Specify the name or IP address of Agent ie., newpc or 132.18.4.12
4. Specify the Community that the Agent belongs to.
5. Specify Object ID.
6. Select required Trap Type from the list options.
7. In Advanced View, specify the SNMP version V1 or V2C.
8. Specify the Timeout required in milliseconds .ie.,1000
9. Specify the Remote port.
10. Specify the number of retry times for the message.
11. Click Save.

Also See...

- SNMP Command

System Command - Examples

Example 1: Lock the computer.

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

2. In the Task Editor, in the Commands list, double-click System. A submenu appears.

3. Double-click Lock Computer. The System dialog appears. Note the green symbol indicating that the Shutdown command can help save energy and spare the environment.

4. Click Save.

Example 2: Shut down the computer.

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

2. In the Task Editor, in the Commands list, double-click System. A submenu appears.

3. Double-click Shutdown. The System dialog appears. Note the green symbol, indicating that the Shutdown command can help save energy and spare the environment.

4. Click Save.

Example 3: Restart the computer.

1. In Automation Anywhere Client, open the Task Editor. To open an existing task, select the task in the Task List, and click Edit. To create a new task, click New, or choose Tools > Task Editor.

2. In the Task Editor, in the Commands list, double-click System. A submenu appears.

3. Double-click Restart.

4. Click Save.

```
Note: Always put System commands last in tasks, since actions entered after the System command
will not be executed.
```

```
Tip : You can have Automation Anywhere Client perform an auto-login, so that when your computer
restarts, Automation Anywhere Client will enter your username and password. Learn more about auto-
login .
```

Also See:

- System Command

Terminal Emulator Command - Examples

Available in Enterprise and Small Business editions only.

Example 1: Obtain ipconfig of remote terminal using default prompt actions.

1. Open the Task Editor.
2. Double-click the Terminal Emulator command. Select the **Connect** option.
3. Specify the *Connection Type*, e.g., TCP/IP
4. Specify *Host Name* of the remote terminal.ie., Admin.
5. Select the *Terminal Type* from the drop down list.
6. The *Port* is set to 23.
7. Insert the *Default Prompt command* of the terminal ie.,C:\Documents and Settings\Admin> cannot be extracted.)
8. Check *Wait For Default Prompt* if you to issue a delay. Specify the *Wait Time in milliseconds*.
9. Check the Show *Terminal Window* if you want to view the action as it happens.
10. Click Save.
11. Select the **Send Text** feature.
12. In the *Send Text* type ipconfig.You can also press F2 to insert a variable. If you wish to encrypt the text, select the *Encrypt Text* check box.
13. Select an option if *Wait for Default* is desired.
14. Click Save.
15. Select the **Get Text** feature.
16. Select *All lines* form *Get Text* option.
17. Insert a variable in *Assign Text To Variable* text box i.e., $Clipboard$. You can also press F2 to insert a variable.
18. Click Save.
19. Select Message Box from the Commands list..
20. Enter a caption title for the *Message box*. You can also press F2 to insert a variable.
21. Insert variable $Clipboard$ in Please *enter message to show the user* text box. Specify scrollbar options for message box if desired.
22. Click Save.
23. Insert a disconnect command from the terminal selecting the **Disconnect** feature .
24. When you run this task the ipconfig of the Admin terminal will be viewed as a message on your computer screen.

Also See...

- Terminal Emulator Command

Variable Operation Command - Examples

Example 1: Use Variable Operation for an array variable for assignment and reinitialize operations.

1. Open the Task Editor.

2. See below task with all details mentioned as comments for execution of task.

1 Comment: We use an array variable to read from existing excel file and then resue it using data from another csv file extracted from website and then insert to database.
2 Connect to "$ConnectStr$" Session:'Default'
3 Comment: ArrayRows is a system defined variable used to retrieve total rows in the user defined array.
4 Comment: ArrayExl - is a user defined array varaible with source as Read from Excel file . Here in our example we use the list of the books and games of the kids having title and cost, which is initalized in Variable Manager.
5 Variable Operation: $ArrayRows($ArrayExl$)$ To Row
6 Start Loop "Row" Times
7   Comment: To skip the header title of the csv extracted from the website use below If condition
8   If $Counter$ Equal To (=) "1" Then
9   Continue
10   End If
11   Execute SQL Statement: 'Insert into Ebaykids (BookTitle,cost) values ("$ArrayExl($Counter$,1)$",'$ArrayExl($Counter$,2)$')' Session: 'Default'
12 End Loop
13 Open "http://www.ebay.com/"
14 Comment: Extracting the books on the philosophy into a csv file using Web Recorder
15 Set text 'philosophy books' into _nkw in the webpage 'Electronics, Cars, Fashion, Collectibles, Coupons and More Online Shopping | eBay'
16 Click on 'Search' Command Button in the webpage 'Electronics, Cars, Fashion, Collectibles, Coupons and More Online Shopping | eBay'
17 Loop While Web Control Exists (Other HTML Control : Next) in the webpage 'philosophy books | eBay'
18   Extract Multiple Data from Webpage to $filePath$ from the web page: philosophy books | eBay
19   Click on 'Next' Other HTML Control in the webpage 'philosophy books | eBay'
20 End Loop
21 Extract Multiple Data from Webpage to $filePath$ from the web page: philosophy books | eBay
22 Comment: Reinitialising the Array-read from excel file (ArrayExl) with the csv file extracted from the website.
23 Variable Operation: Reinitialize $ArrayExl$
24 Variable Operation: $ArrayRows($ArrayExl$)$ To Row
25 Start Loop "Row" Times
26   If $Counter$ Equal To (=) "1" Then
27   Comment: To skip the header title of the csv extracted from the website
28   Continue
29   End If
30   Execute SQL Statement: 'Insert into Ebayphilosophy (BookTitle,cost) values ("$ArrayExl($Counter$,1)$",'$ArrayExl($Counter$,2)$')' Session: 'Default'
31 End Loop

Example 2: Use different list type of variable and perform Assign and Reinitialize Operations on them. Here we are using list variable to then send emails.

1. Open the Task Editor.

2. See below task with all details mentioned as comments for execution of task.

1 Comment: This Loop will execute for 5 times as List read from text file variable has 5 entries.
2 Comment: Create Variable List Type : Source = Read from text file having this RDIsttxt = 11, 21,31,41,51
3 Start Loop "List Variable $RDIsttxt$"
4   Comment: Message box will show the addition of list variable value with other variable value.Here row=10
5   Variable Operation: $RDIsttxt$ + row To $Prompt-Assignment$
6   Message Box: "$Prompt-Assignment$"
7 End Loop
8 Comment: Reinitialize Variable List Type : Source = Read from text file having this RDIsttxt support@automationanywhere.com, testing@gmail.com, sales@automationanywhere.com, buynow@automationanywhere.com
9 Variable Operation: Reinitialize $RDIsttxt$
10 Start Loop "List Variable $RDIsttxt$"
11   Comment: This Loop will execute for 4 times as 'List read from text file variable has 4 entries.
12   Comment: Below command will the send emails to the address ids mentioned in the above Read from text file -List variable.

13   Send Email: Subject "Automation Anywhere finished executing the task." with Attachment(s).

14   Message Box: "Email to '$RDIsttxt$' has been sent sucessfully."

15 End Loop

16 Comment: Create Variable List Type: Source is Value.

17 Comment: In the below Prompt command select more than one file

18 Prompt For File: "Please browse and select multiple files:" for File Assign to variable "$list_val$"

19 Variable Operation: sales@testinganywhere.com To $list_val(2)$

20 Start Loop "List Variable $list_val$"

21   Comment: This loop will execute no. of files selected in Prompt command.

22   Message Box: "$list_val$"

23 End Loop

Example 3: Use yesterday's date as a variable.

1. Open the Variable Manager from Tools > Variable Manager.

2. Click Add new variable.

3. Set the variable type as Value, type a name for the variable (e.g., yesterday), and enter 1 as a value. (This will be the temporary value.)

4. Open the Task Editor.

5. Double-click Variable Operations.

6. On the left side of the = sign, select the variable you created from the drop down list.

7. On the right side of the = sign, press F2 and select the system variable date and click insert.

8. Type -1 after $Date$.

9. Click Save.

Example 4: Convert the value of a variable to thousands.

1. Open the Task Editor.

2. Double-click Variable Operations.

3. For select variable, select the variable whose variable whose value you want to convert into thousands.

4. On the right side of the = sign, press F2 and insert the variable whose value you want to convert into thousands.

5. Type / 1000 after the variable.

6. Click Save.

Example 5: Increment the value of a variable by 1 without using $Counter$.

1. Open the Task Editor.

2. Double-click Variable Operations.

3. For select variable, select the variable whose value you want to increment by 1.

4. On the right side of the = sign, press F2 and insert the variable whose value you want to increment by 1.

5. Type +1 after the variable.

6. Click Save.

Also See...

- Variable Operations Command

Using Web Recorder Command - Examples

Example 1: Extract data from Website using pattern based data extraction.

1. Click New > Web Recorder in the main window.

2. Enter website URL for the website that you wish to extract data based on pattern . eg., an URL of a book listing site.

3. Click Start to begin recording. Please wait until Internet Explorer launches the URL completely.

4. Maximize window.

5. Click Extract Data > Pattern based data

6. Click Add to add the fields that you wish to capture from the selected web page eg., titles of books, author names, publisher, book icons, etc.

7. Click Next.

8. Check *The data spans across multiple pages* box to select similar data from subsequent pages.

9. Click Capture button to select the link that takes you to the next page.

10. Click Next.

11. Click Preview data to preview the data captured.

12. Specify the folder to save images into and specify a file ie., csv to save the extracted data into.

13. Images extracted from multiple pages will be saved in different folders.

Example 2: Extract tabular data from a Web page.

1. Click New > Web Recorder in the main window.

2. Enter website URL **http://www.automationanywhere.com**. You can add the URL to Favourites for easy access again.

3. Click Start to begin recording. Please wait until Internet Explorer launches the URL completely.

4. Maximize window.

5. Click on Purchase > Volume Pricing in the webpage.

6. Select the web control you require in the Web recorder window that opens. Click Save.

7. Click on Extract Table button in the Web Recorder toolbar.

8. Move the yellow prompt bar provided to the table you wish to select, i.e, Feature table.

> Note: The yellow prompt bar will change to red if the content you click on is not a table.

9. A preview of the extracted table will open for your immediate viewing. Save the file as a .csv file.

10. Use the Append to an already existing csv file option, if a CSV file has already been defined .

11. Click OK.

12. Stop recording the task.

13. Save the task.

> Note: When you run this task, it will extract the table and append it into the csv.file if specified.

Wait Command - Examples

Example 1: Wait for a window titled 'Untitled - Notepad' to open.

Open the Task Editor.

1. Double-click the Delay/Wait command.
2. Select Wait For.
3. Select Wait For Window To Open, and type the window title: 'Untitled - Notepad'.
4. Type how long you want Automation Anywhere Client to wait for the condition to be true, i.e., for Notepad to open.
5. You can specify the task to continue or stop if the Notepad window doesn't open with the specified time.

Using Wildcards (*)

1. You can use wildcards (*) in the following options: Wait For Window To Open, Wait For Window To Close, and Wait for Screen Change (relative to Window).
- For example, if you want to perform an operation in any window with the word 'Untitled' anywhere in the title, you can insert a * wildcard character before and after 'Untitled'. In this case, you can specify Wait For Window To Open *Untitled*.
2. Automation Anywhere Client will first search for the exact window named 'Untitled,' and if it does not find it, it will look for windows with 'Untitled' anywhere in the title.
3. You can insert the wildcard character (*) at the beginning or end of the window title.

Example 2: Wait for a submitted query to return successfully on the Web.

1. Open the Task Editor.
2. Click the Delay/Wait command.
3. Double-click Wait For Screen Change.
4. Keep the Web page open to the page where the result of the query will be displayed.
5. Select the relevant option based on whether you want to capture the image relative to the screen, or a particular window. If you select Window, you will be required to select a window from the list.
6. Click Capture and capture a rectangular selection around the area that will change when the query returns.
7. Specify 1 second in the box titled 'How long you want to wait before comparing the screen.'
8. Specify the maximum number of seconds you want the task to wait, and whether you want Automation Anywhere Client to stop the task or proceed to next command.
9. Save the command.

Also See...

- Delay/Wait Command

Web Recorder Command - Examples

Example 1: Extract tabular data from a Web page.

1. Open the Task Editor.

2. Double-click the Web Recorder command and select Extract Table.

3. Specify a URL, e.g., http://www.yourserver.com.

4. Launch the URL.

5. Click Capture and click the table you want to extract.

6. Specify the path of the CSV file where you want the table data to be saved when the task runs.

7. Save the command.

8. After you call the Extract Table command, Automation Anywhere Client stores all the tabular data in a system variable called $Table Column(index)$. You can use this variable to get the rows and columns of the table that you extracted.

9. Double-click the Loop command.

10. Select 'Each row in an Internet Explorer Table'.

11. To access each row and column in the $Table Column$ system variable provided by Automation Anywhere Client, use 'Loop for Each row in an Internet Explorer table' to loop through all the rows of the table. To access each column, use $Table Column(index)$; for example, to get the first column, use $Dataset Column(1)$, and so on.

12. You can use Insert Keystrokes commands with the $Table Column$ variable to type the results into the Excel document.

Example 2: Click a link that changes its location in Internet Explorer.

1. Open the Task Editor.

2. Double-click the Web Recorder command, and select Open Browser.

3. Specify the URL (e.g., http://www.yourserver.com).

4. Save the command.

5. Double-click the Web Recorder command again and select Manage Controls.

6. Specify the URL where the link is present; in this case, the same page you specified in Open Browser.

7. Click Launch. Click Capture and leave the mouse on the link you want to click.

8. The link information will be captured.

9. Save the command.

10. Save the task.

11. When you run the task, it will open the URL and click the link you specified.

Example 3: Download an http file.

1. Open the Task Editor.

2. Double-click the Web Recorder, and select Download File.

3. Specify the download file URL (e.g., http://www.yourserver.com/myfile.exe).

4. Save the command.

5. Specify a local filename to store on your machine.

6. Save the command.

Example 4: Check all links in a Web page.

1. Open the Task Editor.

2. Double-click the Web Recorder command and select Find Broken Links.

3. Specify the URL (e.g., http://www.yourwebpage.com).

4. Select Test Specified Webpage Only.

5. Specify the local path for the CSV file to store the results of the command.

6. Save the command.

7. When you run the command, it will test all links in the Web page you specified, and store the results in the CSV file you named.

Example 5: Check all links in a website.

1. Open the Task Editor.

2. Double-click the Web Recorder command, and select Find Broken Links.

3. Specify the website address (e.g, http://www.yourwebsite.com).

4. Select Test Entire Website.

5. Specify a local file path for the CSV file to store the results of the command.

6. Save the command.

7. When you run the command, it will test all the links in the website you specified, and the results will be stored in the CSV file you named.

Example 6: Always click the third link in the CNN Headlines.

1. Click Tools > Web Recorder.

2. Type http://www.cnn.com and click Start

3. Click the third link, then click Stop recording.

4. Open the task in the Task Editor.

5. Edit the Click command.

6. Un-check the Name check box, and check the Index check box. Now, every time Automation Anywhere Client reaches this command it will search the link by its index (and not the name or URL it points to). Thus, even if the name of the link changes, it will click the third link.

Example 7: Extract data from a website and paste it into a blank Word document.

1. Open the Task Editor.

2. Double-click the Web Recorder command.

3. Double-click Extract Data.

4. Type a website URL and click Launch (e.g., http://www.automationanywhere.com/products/automation-software.htm).

5. Once the site is open, you are redirected to the Extract Data dialog box.

6. Click Capture.

7. The mouse appears as yellow background text indicating Please click on the data you want to extract.'

8. Move the mouse over the text you want to copy, and click anywhere in the text. (If the text background of the mouse changes to red, it means the text is not available for copying.)

9. Assign the copied text value to a variable from the list; e.g., $my-variable$.

10. Click Save.

11. Make sure a blank Word document is open.

12. Double-click Insert Keystrokes.

13. Select the window title from the list.

14. Insert the variable from the list on the right side of the dialog box, i.e., $my-variable$.

15. Click Save.

```
Tip : You can specify keywords that determine the range of your text. For example, from the text
'Your reference no. is 245698 for today,' if you want to copy only the number and avoid the extra
text, you can specify 'is' as the 'Before Keyword' and 'for' as the 'After Keyword.'
```

Also See

- Web Recorder Command

Web Service Command - Examples

Available in Enterprise and Small Business editions only.

Example 1: Creating a Web service.

1. Open Web Service command.

2. Enter complete URI for any web service which is up and running.

- Example:http://www.xignite.com/xGlobalQuotes.asmx?WSDL

3. Click the Test Output button to view the Web Service Output data.

4. Save the data as an XML file or assign it to a variable.

5. Save and exit the Web Service command.

Also See...

- Web Service Command
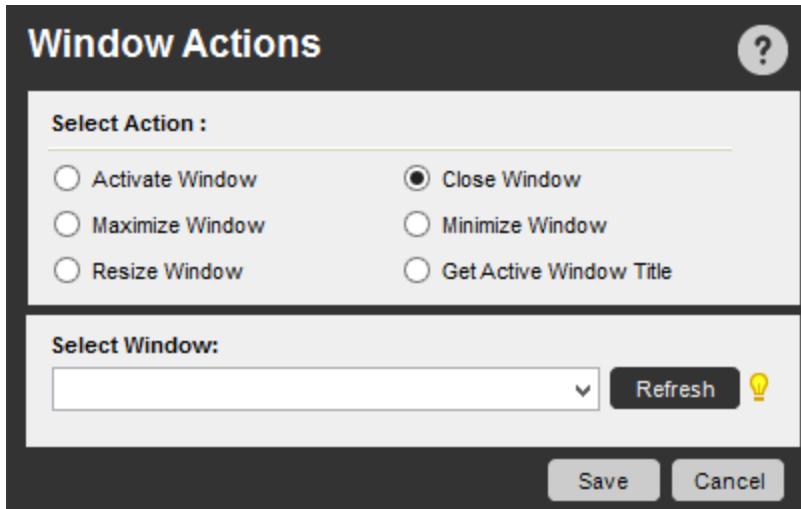
Window Actions Command - Examples
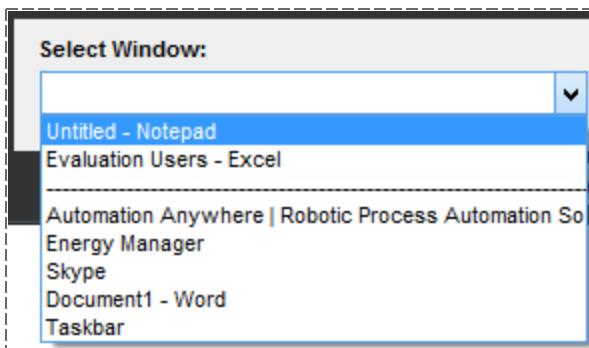
Use Case Scenario 1

Close the window 'Untitled - Notepad' in Microsoft Windows Notepad.

1. Start Microsoft Notepad. Notice that the title bar says 'Untitled - Notepad.'

2. In Automation Anywhere Client, open the Task Editor.

3. In the Task Editor, in the Commands list, double-click Window Actions. A sub-menu appears.

4. Double-click Close Window. The Window Actions dialog appears.

5. In the Select Window drop-down, select 'Untitled - Notepad.'

6. Click Save.

Use Case Scenario 2

Close all open windows.

1. In Automation Anywhere Client, open the Task Editor.

2. In the Task Editor, in the Commands list, double-click Window Actions. A sub-menu appears.

3. Double-click Close Window. The Window Actions dialog appears.

4. From the Select Window drop-down, select 'Close All Open Windows.'
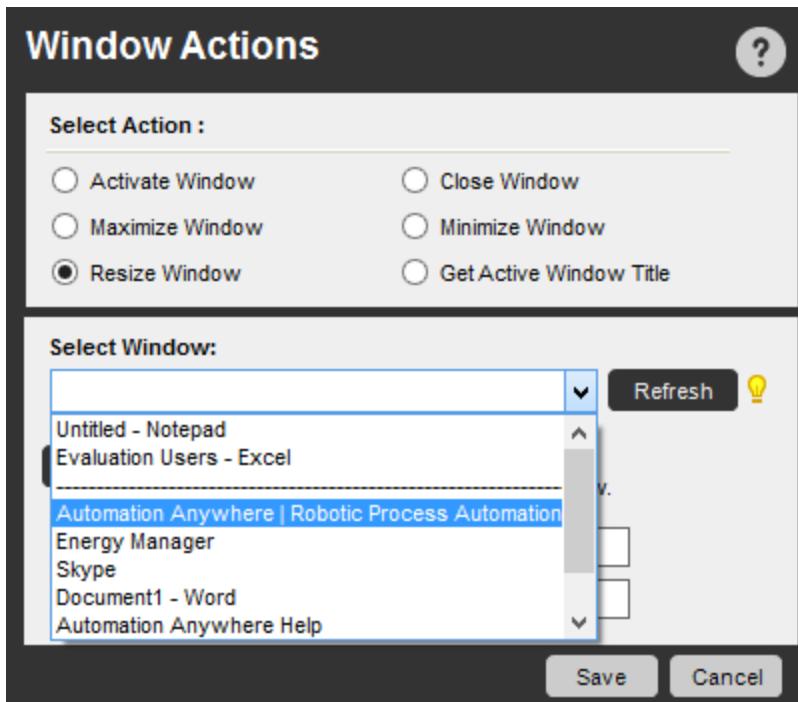
5. Click Save.

Use Case Scenario 3
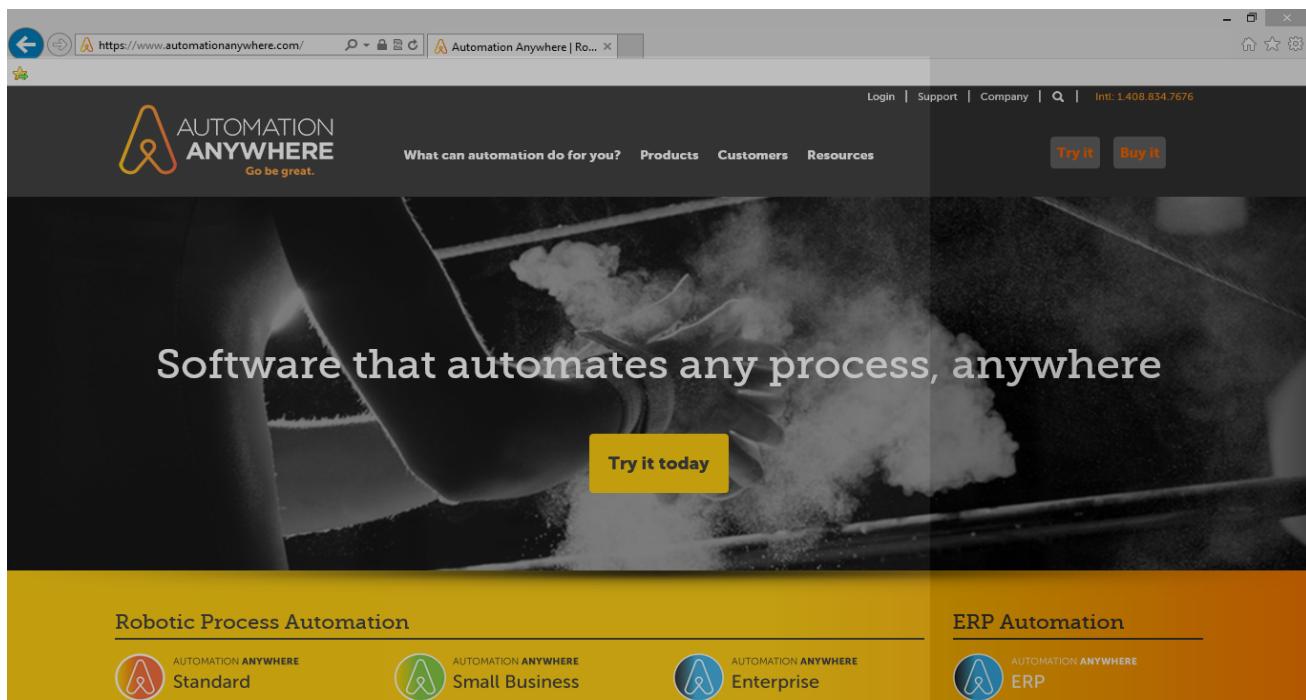
Re-size a browser window

1. Open a Web page in a browser.
2. In Automation Anywhere Client, open the Task Editor.
3. In the Task Editor, in the Commands list, double-click Window Actions. A sub-menu appears.
4. Double-click Re-size Window. The Window Actions dialog appears.
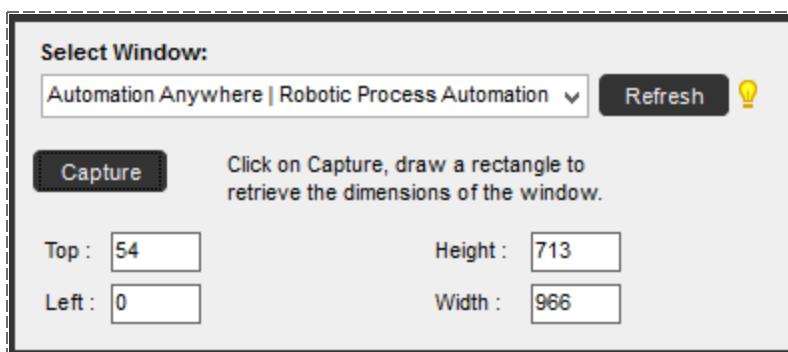
5. From the Select Window drop-down, select the browser window title that you want to re-size.

6. Click Capture, and draw a rectangle of the desired size.

7. This will retrieve the coordinates for the re-sized window.
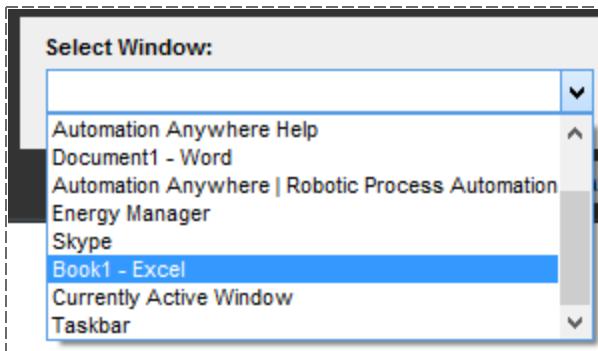
8. Click Save.

Use Case Scenario 4

Maximize a Microsoft Excel file named 'Microsoft Excel-Book1.'

1. Open the file Book1.xls in Microsoft Excel.
2. In Automation Anywhere Client, open the Task Editor.
3. In the Task Editor, in the Commands list, double-click Window Actions. A sub-menu appears.
4. Double-click Maximize Window. The Window Actions dialog appears.

5. From the Select Window drop-down, select the window titled 'Microsoft Excel-Book1.xls.'
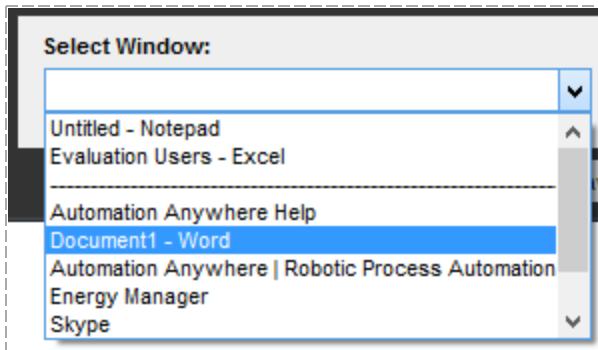
6. Click Save.

Use Case Scenario 5

Activate a Microsoft Word document titled 'Document1.'

1. In Microsoft Word, open a new document - Word automatically names it Document 1.

2. In Automation Anywhere Client, open the Task Editor.

3. In the Task Editor, in the Commands list, double-click Window Actions. A sub-menu appears.

4. Double-click Activate Window. The Window Actions dialog appears.

5. From the Select Window drop-down, select 'Document1-Microsoft Word.'

6. Click Save.

Also See...

- Web Actions Command

XML Command - Examples

Feature also available in Standard edition version 8.2 or higher.

Example 1: Get XML data by defining nodes and saving the XML data at the required location.

1. Start XML Session using an existing file, for eg.,"C:\XML\Bookstore.xml". Keep the Session as Default.

2. Update the Node value to "2010" for a XPath location "//book[3]/year".

3. Insert Node name and Node value as "newNode" and "newVal" respectively for XPath location "//book[3]/year".
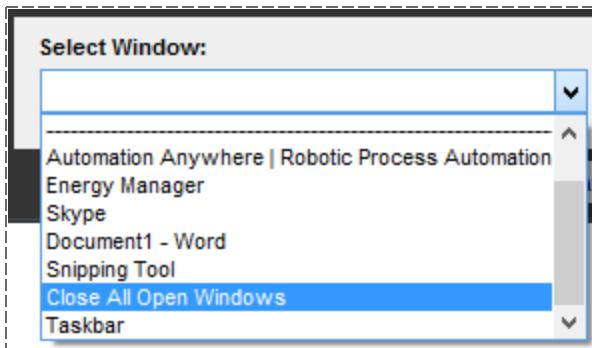
4. You can execute a XPath Function "sum(//book/price)" and assign the returned value to a variable.

5. To get text value from multiple nodes given XPath expression "//book/title", go to Get Node(s) > Multiple Nodes.

6. Start Loop for 'Each node in an XML datasheet'

7. Insert a comment and prompt a message box at this juncture.

8. End Loop

9. Save the session data assigned to the variable and write the Session Data into a new file at the required location. Overwrite the file if it exists.

10. End the session by selecting End XML session

Also See...

- XML Command

Using Variables

Why Use Variables?

Variables are storage locations for known or unknown information.

When building automated tasks, variables play an important role in maintaining or calculating information.

Variables can help you in a number of ways, from fetching online data to transferring data between applications, such as Microsoft Excel.
Automation Anywhere is designed with various types of variables which can be defined for each task.
 These are known as local variables. It has pre-defined system variables available for all tasks.

The topics in this section will help you to understand the types of variables, how to use them, and when to use them.

When to Use Variable?

As you build automation tasks, the scripts that are generated can lend themselves to using variables to track or modify the value of a field or data point. For example, if you want to loop through a set of data in a CSV file, you can use a variable to copy the first row, paste it to another application, move to the second row, and so on.

All actions that you take in a task can support the use of variables. These actions can include:

- Selecting windows
- Opening programs
- Working with files, folders, keystrokes, spreadsheets, loops, and more.

In This Section
- Types of Variables
- Working with Variables

Knowing When to Use Variables

This topic explains when you might want to use variables in your automation tasks.

As you build automation tasks, the scripts that are generated can lend themselves to using variables to track or modify the value of a field or data point. For example, if you want to loop through a set of data in a CSV file, you can use a variable to copy the first row, paste it to another application, move to the second row, and so on.

All actions that you take in a task can support the use of variables. These actions can include:

- Selecting windows
- Opening programs
- Working with files, folders, keystrokes, spreadsheets, loops, and more.

Two special cases of using variables include:

- Passing a variable between Running Tasks
- Using variables with IF/Else and LOOP commands

Also See...

- Understanding the Types of Variables
- Using the Variable Manager
- Creating a New Variable
- Assigning Variables to a Task
- Reinitializing Variables
- Resetting System Variables
- Reading Variables from a Text File
- Watching Variables

Passing a Variable between Running Tasks

You can pass an individual variable value from one task to another by using this option. The value of the variable in the first running task becomes the input for the variable in a second task. This feature is important when tasks are run in nested manner.

Tip: The same type of variable should be defined in both tasks.

To pass a variable between running tasks, follow these steps:

1. Select the Run Task command and select the Variable option.

2. In the "Create variable association" field, select the variable from list in Main Task that has to be passed on to the consequent 'Run Task' variable.

Also See...
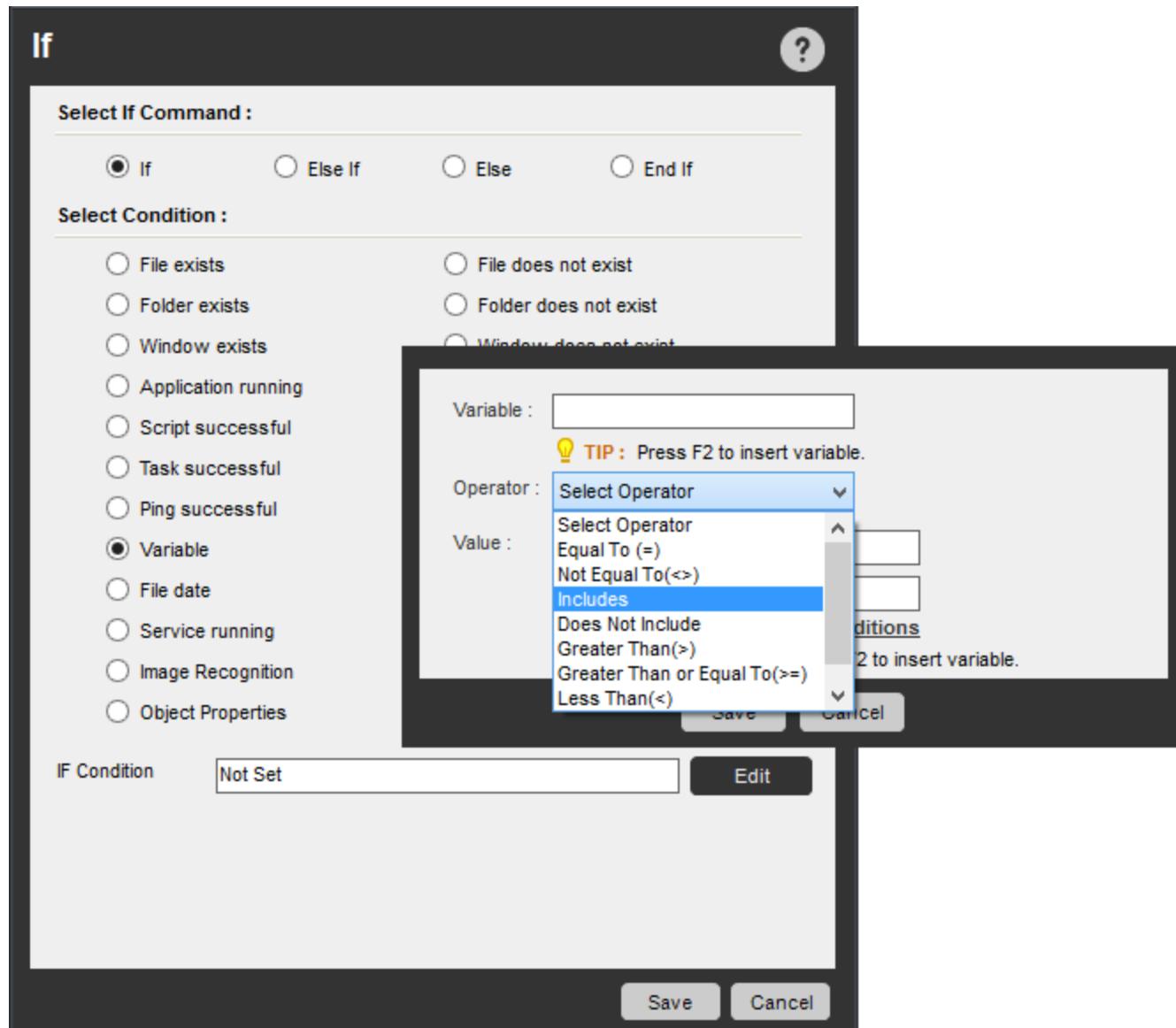
- Using Variables with IF-Else and LOOP Commands

Using Variables with IF-Else and LOOP Commands

The IF/Else and LOOP commands enable you to use variables, allowing you to run a loop until a particular condition is met for the variable.

Using IF/Else Commands with Variables

While inserting an If/Else condition, select a variable you want to use (see example).

Using Loop Commands with Variables

While inserting a Loop condition, select a variable you want to use within the loop until the condition is met (see example).

Other User
Resources:

User forums

Ask the Expert

Open a support case

Automation Examples

Also See...

- Passing a Variable between Running Tasks

Types of Variables

This topic describes the types of variables you can define and use when building automated tasks.

Two types of Variables exist in Automation Anywhere tasks: user (local) variables and system variables.

User (Local) Variables

User variables are typically defined by an automation user, and used for a particular task or set of tasks. The purpose of a user variable can range from simply holding a single value to holding multiple values, as would be used in an Excel spreadsheet.

For new automation tasks, two variables are pre-defined for your use:

- my-list-variable (type: List)
- Prompt-Assignment (type: Value)

After you create a user (local) variable, you can reinitialize it or assign it when a task runs.

System Variables

System variables are pre-defined variables that are provided by Automation Anywhere. System variables are available for use in all automation tasks.

> 📝 Note: For all Automation Anywhere commands, user and system variables are supported in fields that have the light bulb 💡 symbol present.

Use F2 function key and the list of variables will appear, to select and insert them.

Also See...

- Using the Variable Manager
- Creating a New Variable
- Assigning Variables to a Task
- Reinitializing Variables
- Resetting System Variables
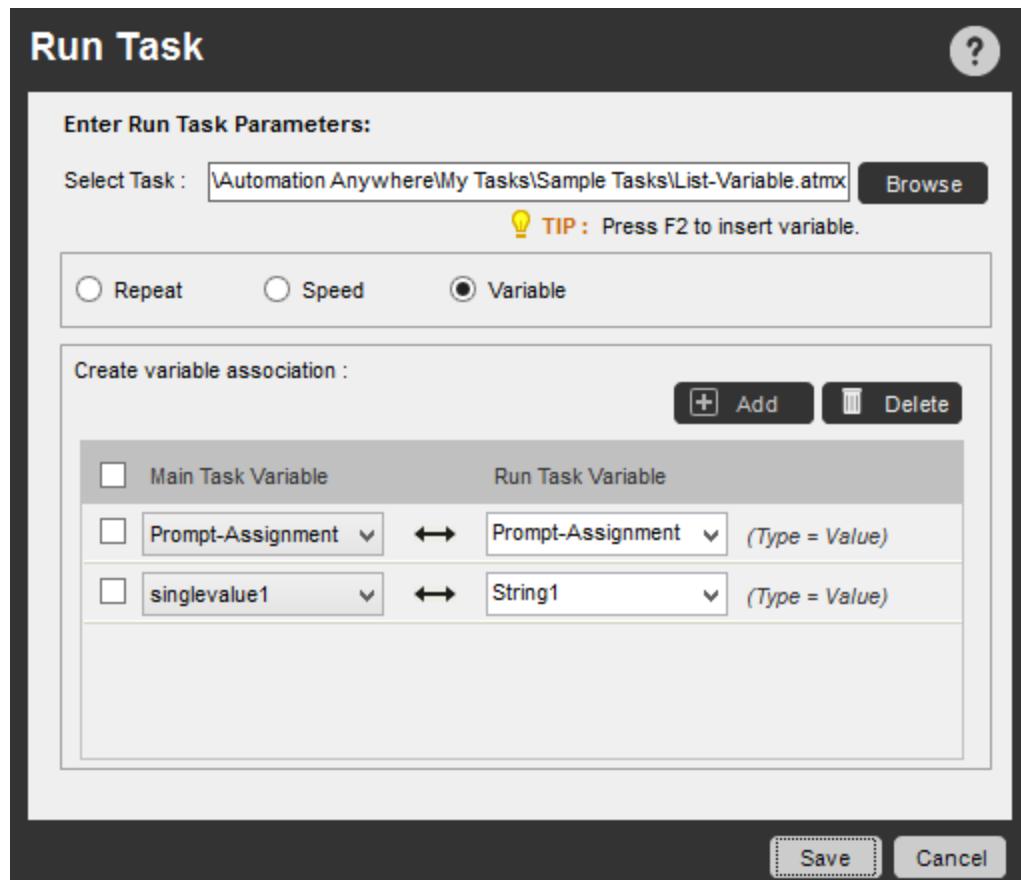- Reading Variables from a Text File
- Watching Variables

Local (User) Variables

User Variables, also known as *local variables*, are those variables that are defined by an Automation Anywhere user for a particular task. All user variables are displayed in the Variable Manager for the respective tasks under 'Local Variables'.

Each user variable has a *source* . This source varies from direct assignment to reading values from files, such as text files, Excel or CSV files, or a database.

Types of user (local) variables:

- Value - Holds a single value.
- List - Holds multiple values in a single dimension (1 X n).
- Array - Holds multiple values in two dimensions (n X n).
- Random- Holds a string or numeric value that is determined by the system while running the task.

> Note: Local Variables can be created only if you have a developer license.

Also See...

- Using the Variable Manager
- Creating a New User Variable
- Assigning Variables to a Task
- Reinitializing Variables
- Resetting System Variables
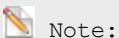
Value Type Variables

When/Why To Use a Value Type Variable

You can use a value type variable when you need to hold a single data point and use it in multiple places. This "placeholder" value can represent either text or numeric data.

After you create the variable, you can use it by inserting the variable in several of the Task Editor commands. When the value of the variable is modified, this value is reflected in any subsequent commands that are run by the task.

You can also mark a value type variable as 'Regular Expression' . These, you can use when creating tasks that require pattern based searches in files, folders and window title commands.

```
Note:
  • You can input only the first line of text in the 'Value' text box.
  • Regular Expression in variables can be used in commands that support wild-cards. For instance,
    you can use regular expressions in certain sub-commands of Files/Folders; namely Copy, Delete,
    Zip and Print Multiple Files/Folders.
```

For a list of commands that support variables with regular expression, refer the section 'Commands that support Regular Expression'.

How to Create a Value Type Variable

You can create value type variables in two ways: using direct assignment or using a file.

Direct Assignment

Create a value type variable using direct assignment. This is used when values are generally known and needs to be retrieved repetitively for different commands in the task. This makes the task flexible and easy to use.

To create a value type variable using direct assignment, follow these steps:

1. In the Task Editor, click on the Variable Manager icon at the top or on the tab on the right side.
2. Click on the Add button. The "Add Variable" window is displayed, with the "Create New Variable" option selected.
3. Select type "Value". Generally, this is the default selection.
4. Enter a name for the variable. The name must begin with an alphabetic character and cannot contain spaces.
5. Select the "Value" radio button. This is the default.
6. Specify an initial value. This value can be character or numeric.
7. Click Save.

After the variable is saved, it is displayed in the Local Variables section of the Variable Manager.

Examples of Value Type Variables:

- Name: *ValueDB*, Value: `Oracle`

Add Variable ?

Select Action :
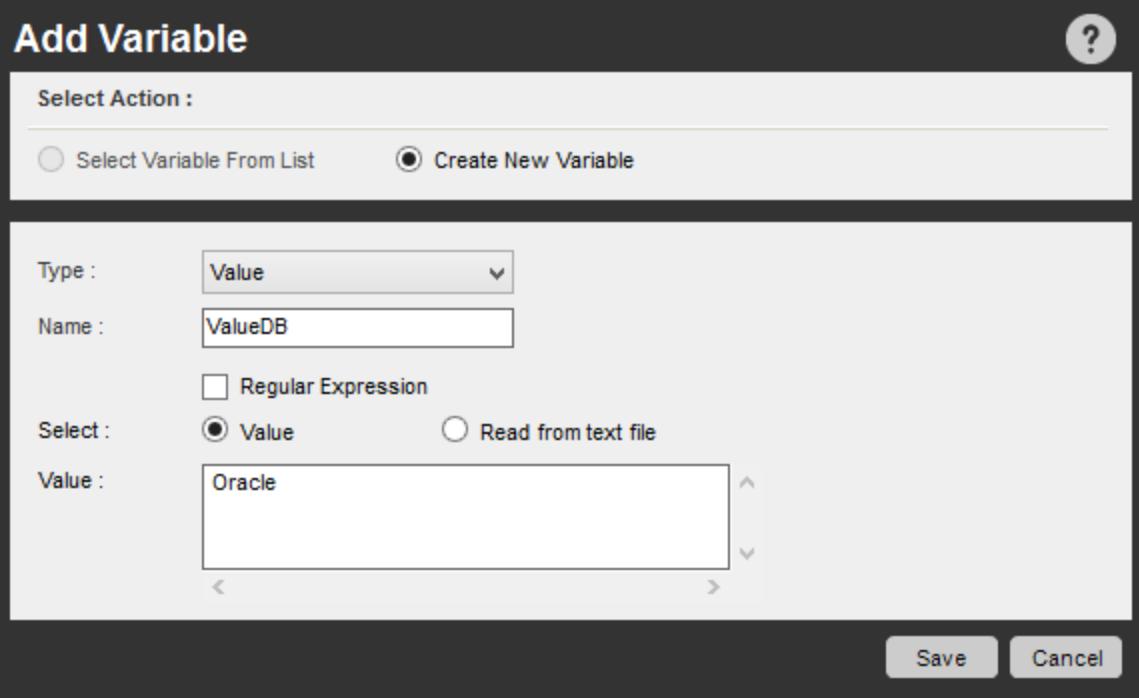
○ Select Variable From List   ● Create New Variable

Type :	Value ▾
Name :	ValueDB

☐ Regular Expression

Select :   ● Value   ○ Read from text file

Value :   Oracle

Save   Cancel

- Name: *email*, Value: `contact@automationanywhere.com`

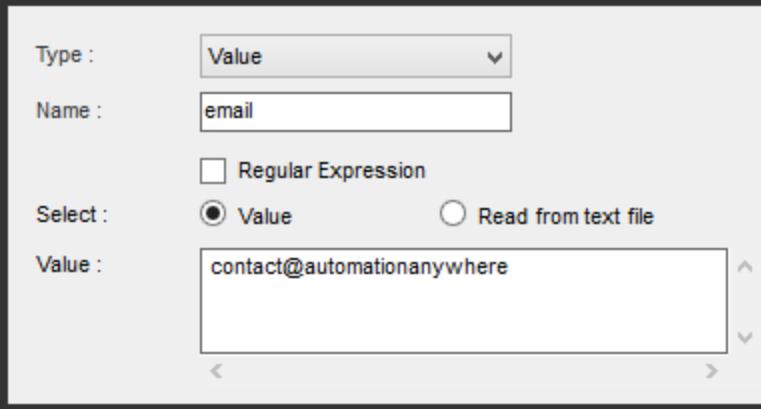|	
Type :	Value ▾
Name :	email

☐ Regular Expression

Select :   ● Value   ○ Read from text file

Value :   contact@automationanywhere

- Name: *Pathname*, Value: `C:\Automationanywhere\`
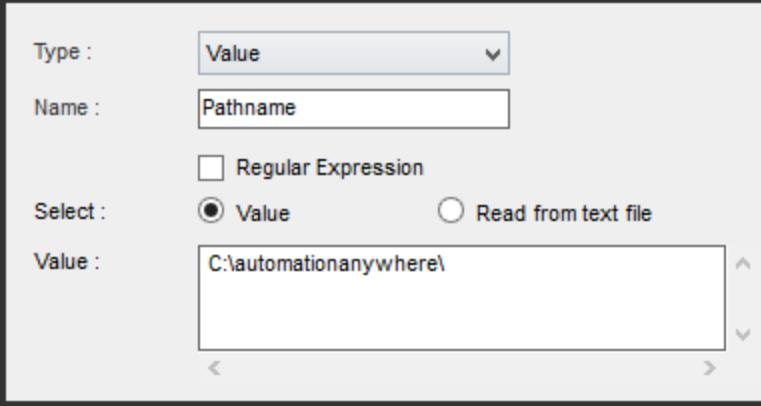
Type :	Value ▾
Name :	Pathname
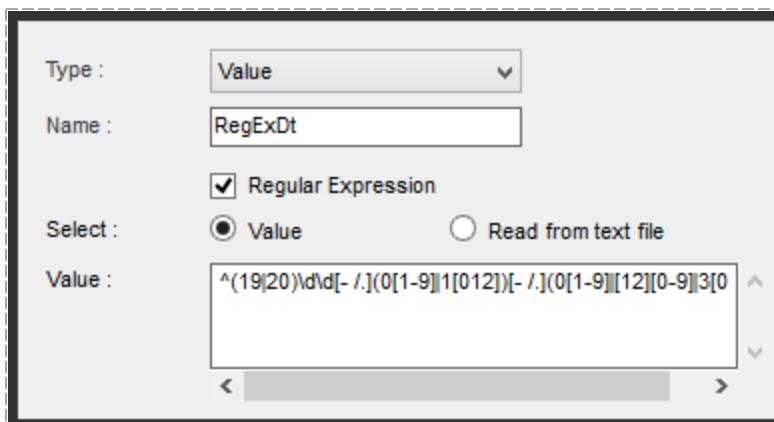
☐ Regular Expression

Select :   ● Value   ○ Read from text file

Value :   C:\automationanywhere\

- Name: *RegExDt*, Value: `^(19|20)\d\d[- /.](0[1-9]|1[012])[- /.](0[1-9]|[12][0-9]|3[01])$`

Common examples of names you can use for value type variables include:

`Pathname, CounterValue, DatabaseName, email,` etc.

File Assignment
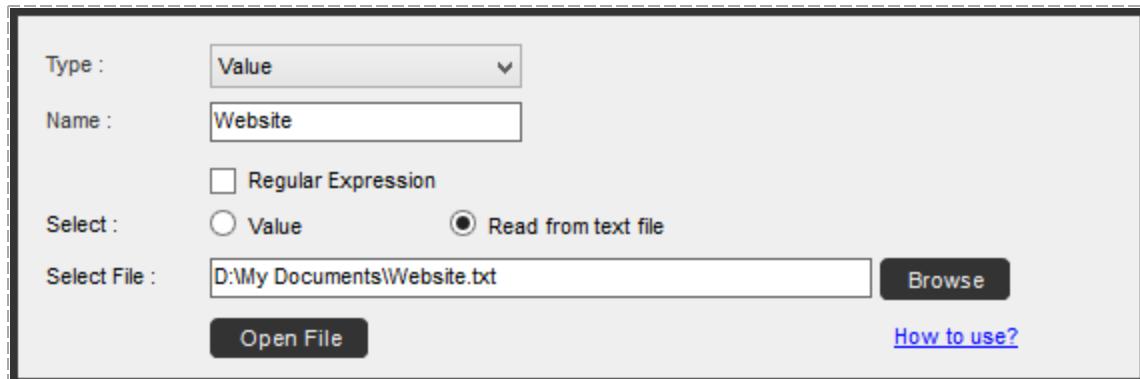
Create a value type variable using file assignment when the value of the variable needs to be retrieved from a text file while a task is running.

Your task can read the file and change the value of the variable, reassigning the value at each data point in the file.

To create a value type variable using file assignment, follow these steps:

1. Determine the text file you will use for assigning values. In the text file, ensure that the variable name exists, followed by an '=' operator. This ensures correct assignment of values. Example: Variable Name: *Website* Text file content: website = auto-mationanywhere.com

2. In the Task Editor, click on the Variable Manager icon at the top or on the tab on the right side.

3. Click on the Add button. The "Add Variable" window is displayed, with the "Create New Variable" option selected.

4. Select type "Value". Generally, this is the default selection.

5. Enter a name for the variable. The name must begin with an alphabetic character and cannot contain spaces.

6. Select the "Read from text file" radio button.

7. In the Select File field, browse to the file or type the file path for the required text file. Use the Open File button to view the selected text file.

8. Click Save.

After the variable is saved, it is displayed in the Local Variables section of the Variable Manager.

Commands that support regular expression

- App Integration
- Delay/Wait
- File/Folder
- Image Recognition
- Insert Keystrokes
- Insert Mouse Click
- Manage Window Controls
- OCR
- Screen Capture
- Window Actions

Also See...

- List Type Variables
- Array Type Variables
- Random Variables
- Pre-defined variables

List Type Variables

When/Why To Use a List Type Variable

You can use a list type variable when you need to retrieve multiple values, one by one. It is basically one dimensional placeholder for data.
 Common uses of list variables include:

- Sending email to multiple recipients
- Passing different values inside of a loop
- Searching multiple web addresses

The values can represent either text or numeric data.

After you create the variable, you can use it by inserting the variable in several of the Task Editor commands. When the value of the variable is modified, this value is reflected in any subsequent commands that are run by the task.

How to Create a List Type Variable

You can create list type variables in two ways: using direct assignment or using file assignment.

Direct Assignment

Create a list type variable using direct assignment when the values of the variable are fixed and need to be retrieved one-by-one as the task runs.

To create a list type variable using direct assignment, follow these steps:

1. In the Task Editor, click on the Variable Manager icon at the top or on the tab on the right side.
2. Click on the Add button. The "Add Variable" window is displayed, with the "Create New Variable" option selected.
3. Select type "List".
4. Enter a name for the variable. The name must begin with an alphabetic character and cannot contain spaces.
5. Select the "Value" radio button. This is the default.
6. Specify list values in the field provided. The values can be character or numeric.
7. Click the "Add to List" button.
8. Repeat steps 5 and 6 to complete the list.
9. Click Save.

You can use the "Make Random" checkbox to call the values randomly when the task runs.

If you leave the option unchecked, the values will be retrieved in the order you have entered them.

After the variable is saved, it is displayed in the Local Variables section of the Variable Manager.

Example of List Type Variable:

Name : *Email Address*

Values: products@automationanywhere.com, support@automationanywhere.com, john.smith@automationanywhere.com

Add Variable

Select Action :

○ Select Variable From List   ● Create New Variable

Type :	List ⌄	List Value :	john.smith@automationanywhere.co
Name :	EmailAdress		products@automationanywhere.cor
			support@automationanywhere.com

☐ Make Random

Select : ● Value

○ Read from text file

[Add To List] [Edit] [Delete]

[Save] [Cancel]

File Assignment

Create a list type variable using file assignment when the values for the variable need to be retrieved from a text file while a task is running.

The text file is used to define the list variable. Loop commands make it easy to process the list of values.

Common Use Case: Extract data to a text file

To create a list type variable using file assignment, follow these steps:

1. Determine the text file you will use for assigning values.

   In the text file, ensure that the variable name exists, followed by an '=' operator, and a list of comma-separated values.

   Example text file with list of values: Variable Name: *products*

   The `products.txt` file contains:

   ```
   products = automation, testing, PI, SAP-automation, ERP-automation
   ```

2. In the Task Editor, click on the Variable Manager icon at the top or on the tab on the right side.
3. Click on the Add button. The "Add Variable" window is displayed, with the "Create New Variable" option selected.
4. Select type "List".
5. Enter a name for the variable. The name must begin with an alphabetic character and cannot contain spaces.
6. Select the "Read from text file" radio button.
7. In the Select File field, browse to the file or type the file path for the required text file. Use the Open File button to view the selected text file.
8. Click Save.

After the variable is saved, it is displayed in the Local Variables section of the Variable Manager.

Add Variable

Select Action :

○ Select Variable From List   ◉ Create New Variable

Type : | List ▼ |

Name : | Products |

☐ Make Random

Select :   ○ Value   ◉ Read from text file

Select File : | D:\My Documents\products.txt | [Browse]

[Open File]   How to use?

[Save]   [Cancel]

Also See...

- Value Type Variables
- Array Type Variables
- Random Variables
- Pre-Defined User Variables

Array Type Variables

When/Why To Use an Array Type Variable

An array variable is a two-dimensional variable that holds multiple values in a table of rows and columns. Arrays are very powerful for creating staging areas for data that needs to be retrieved by your process as it runs.

Common uses of array variables include:

- Extracting data from web pages
- Extracting many rows of data from an Excel spreadsheet or a database
- Read or write data from/to a legacy system, an ERP system, or another application
- Filling out order forms with different fields from Excel to Database

The values can represent either text or numeric data.

After you create the variable, you can use it by inserting the variable in several of the Task Editor commands. When the value of the variable is modified, this value is reflected in any subsequent commands that are run by the task.

How to Create an Array Type Variable

You can create an array type variable in four ways:

- Using direct assignment
- Using a text file
- Using an Excel or CSV file
- Using a database

Direct Assignment

Create an array type variable using direct assignment when you want the values of the variable to be defined directly to the variable.

To create an array type variable using direct assignment, follow these steps:

1. In the Task Editor, click on the Variable Manager icon at the top or on the tab on the right side.
2. Click on the Add button. The "Add Variable" window is displayed, with the "Create New Variable" option selected.
3. Select type "Array".
4. Enter a name for the variable. The name must begin with an alphabetic character and cannot contain spaces.
5. Select the "Value" radio button. This is the default.
6. Specify the number of rows and columns in the field provided. Default values are 1 X 1.
7. Click the Initialize Values button. The Array Value Details window is displayed.
8. Enter the values for each cell.
9. Click the Save button to save the values.
10. Click the Save button again to save the array variable.

You can modify the array dimensions using the Add Row, Add Column, Delete Row, and Delete Column buttons.

After the variable is saved, it is displayed in the Local Variables section of the Variable Manager.

Add Variable

Select Action :

○ Select Variable From List   ● Create New Variable

Type :	Array
Name :	BookMaster
Select :	Value
Row(s) :	5	Column(s) :	2	Initialize Values

Save   Cancel

Array Value

	1	2
1	McGraw Hill	750
2	Science Fiction	1252
3	Autobiography	1200
4	Pop Culture	
5		

Add Row   Add Column   💡 **TIP :** Double click on any cell to update the value of that cell

Delete Row   Delete Column

Save   Cancel

Text File Assignment

Create an array type variable using text file assignment when you want to read values from a text file into the array variable while the task is running. The text file is used to define the array variable.

Loop commands make it easy to process the list of values.

User Scenario: Entering series data into a system.

To create an array type variable using text file assignment, follow these steps:

Determine the text file you will use for assigning values. The values in the text file are retrieved as follows:

Comma-separated values = Column data

Returns (new line) = Row data

1. A variable name is not required inside the text file for array data retrieval.

Example text file with array values:

Variable Name: fruit-prices The fruit-prices.txt file contains:
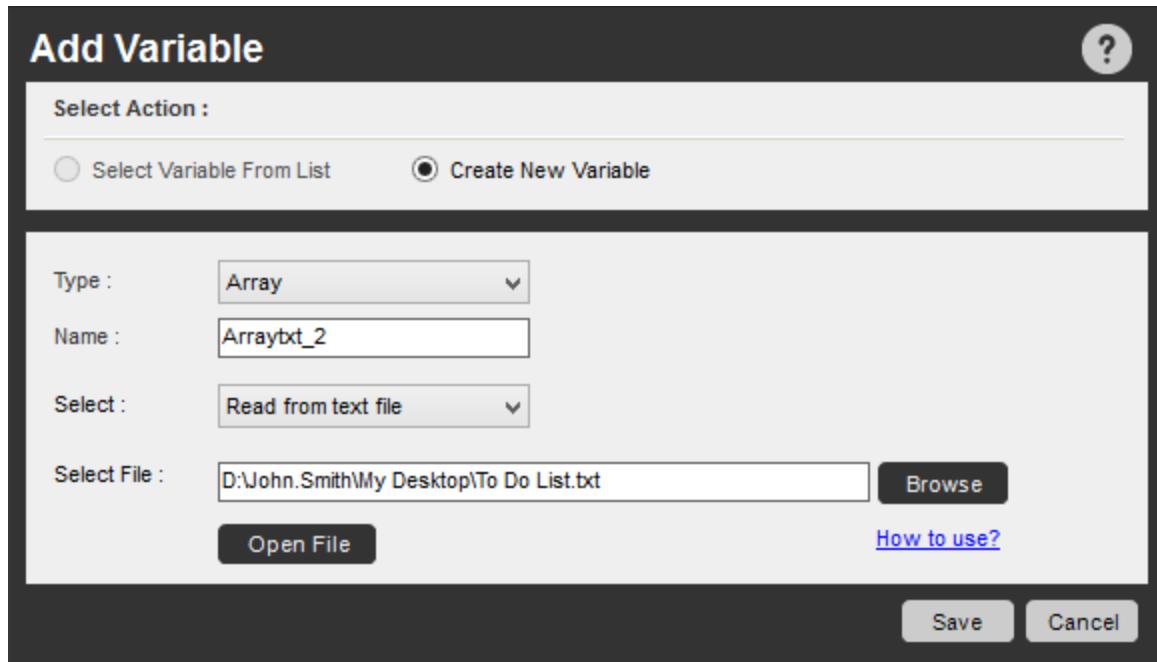
apple,banana,orange,lime,lemon

2.99,0.69,3.14,1.99,0.99

looks like this:

| apple | banana | orange | lime | lemon |

| 2.99 | 0.69 | 3.14 | 1.99 | 0.99 |

----

2. In the Task Editor, click on the Variable Manager icon at the top or on the tab on the right side.
3. Click on the Add button. The "Add Variable" window is displayed, with the "Create New Variable" option selected.
4. Select type "Array".
5. Enter a name for the variable. The name must begin with an alphabetic character and cannot contain spaces.
6. Select the "Read from text file" radio button.
7. In the Select File field, browse to the file or type the file path for the required text file. Use the Open File button to view the selected text file or to modify it.
8. Click Save.

After the variable is saved, it is displayed in the Local Variables section of the Variable Manager.

Excel or CSV File Assignment

The Automation Anywhere Web Recorder and Excel commands extract data to Excel and CSV formats. You can use these files to create an array variable that might be used for entering or manipulating data for computing or analysis.
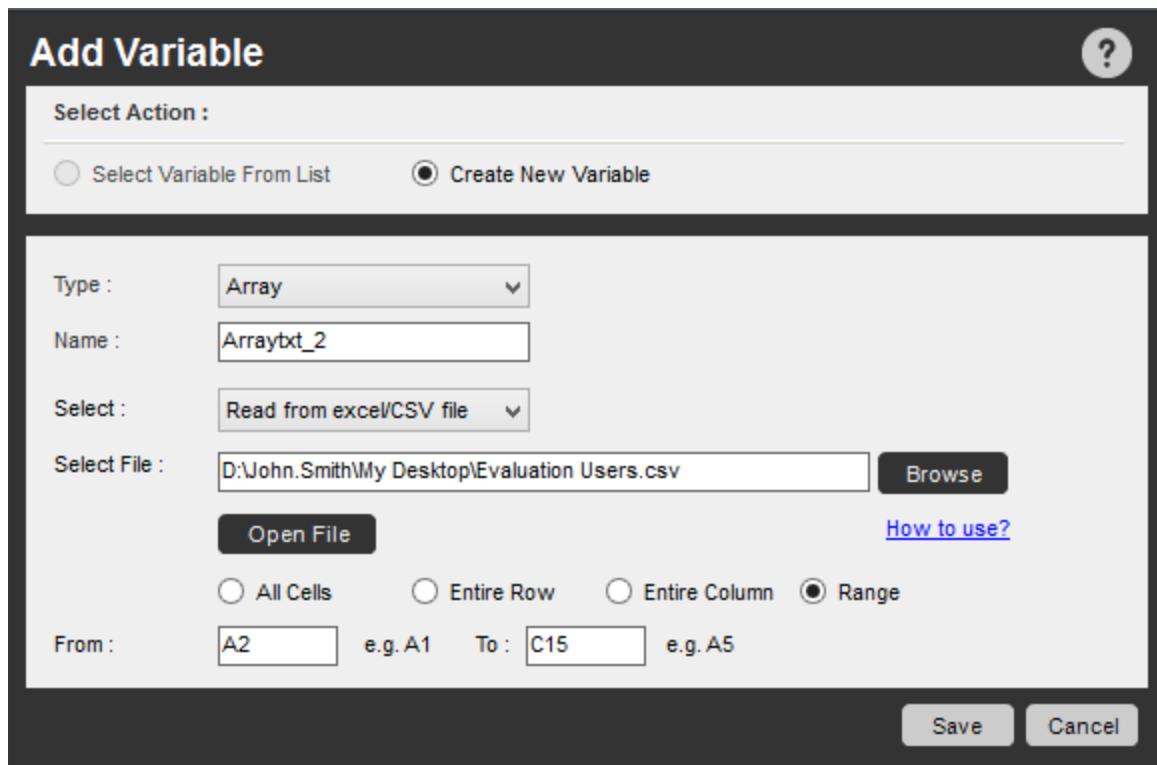
Automation Anywhere provides a range of options for retrieving subsets or partial sections of the data.

Create an array type variable using an Excel or CSV file assignment when you want the values of the variable to be retrieved from an Excel or CSV file.

To create an array type variable using Excel or CSV file assignment, follow these steps:

1. In the Task Editor, click on the Variable Manager icon at the top or on the tab on the right side.
2. Click on the Add button. The "Add Variable" window is displayed, with the "Create New Variable" option selected.
3. Select type "Array".
4. Enter a name for the variable. The name must begin with an alphabetic character and cannot contain spaces.
5. Select the "Read from Excel/CSV file" radio button.
6. Browse and select the required file.
7. Select the options you want for reading values from specific locations in the file.
8. Select All cells, Entire Row, Entire Column, or Range.
   - If you select Range, specify the From and To range. Use the Open File button to view the selected text file or to modify it.
9. Click Save.

After the variable is saved, it is displayed in the Local Variables section of the Variable Manager.

Using a Database

You can define an array variable with values derived from the result of a database query. The array variable is created with dimensions that are determined as a result of running the query.

User Scenario: Extracting data from a database for use in another application or for analysis

Create an array type variable using a database query when you want the values of the variable to be retrieved from a database.

To create an array type variable using a database query, follow these steps:

1. In the Task Editor, click on the Variable Manager icon at the top or on the tab on the right side.
2. Click on the Add button. The "Add Variable" window is displayed, with the "Create New Variable" option selected.
3. Select type "Array".
4. Enter a name for the variable. The name must begin with an alphabetic character and cannot contain spaces.
5. Select the "Read from database" radio button.
6. Specify the database connection string details using Browse.
7. Enter Select Query by typing the entire database query.
8. Click Save.

After the variable is saved, it is displayed in the Local Variables section of the Variable Manager.

Also See...

- Value Type Variables
- List Type Variables
- Random Variables
- Pre-Defined Variables

Random Variables

You can create two types of random variables: string and numerical.

Random variables are useful when you need to generate a random, repetitive string or numerical set. The values are generated when you run the task.

Common use cases:

- String: Generating test data as input for fields or forms
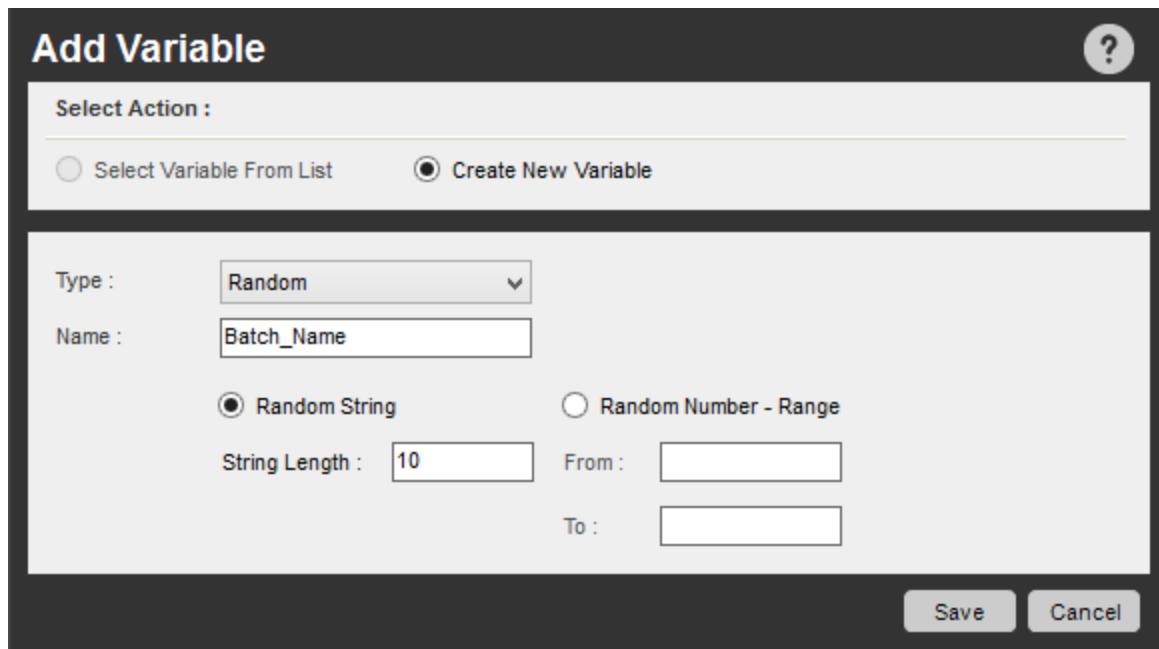- Numerical: Generating ID numbers in batch

After you create the variable, you can use it by inserting the variable in several of the Task Editor commands.

How to Create a Random Type Variable

Creating a Random String Variable

To create a random string variable, follow these steps:

1. In the Task Editor, click on the Variable Manager icon at the top or on the tab on the right side.
2. Click on the Add button. The "Add Variable" window is displayed, with the "Create New Variable" option selected.
3. Select type "Random".
4. Enter a name for the variable. The name must begin with an alphabetic character and cannot contain spaces.
5. Select the "Random String" radio button. This is the default.
6. Specify the string length in the field provided. The value cannot exceed 255.
7. Click Save.

Creating a Random Numeric Variable

To create a random numeric variable, follow these steps:

1. In the Task Editor, click on the Variable Manager icon at the top or on the tab on the right side.
2. Click on the Add button. The "Add Variable" window is displayed, with the "Create New Variable" option selected.
3. Select type "Random".
4. Enter a name for the variable. The name must begin with an alphabetic character and cannot contain spaces.
5. Select the "Random Number - Range" radio button.

6. Specify the "From" and "To" values for the range.

7. Click Save.

Also See...

- Value Type Variables
- Array Type Variables
- List Type Variables
- Pre-Defined Variables

Pre-Defined User Variables

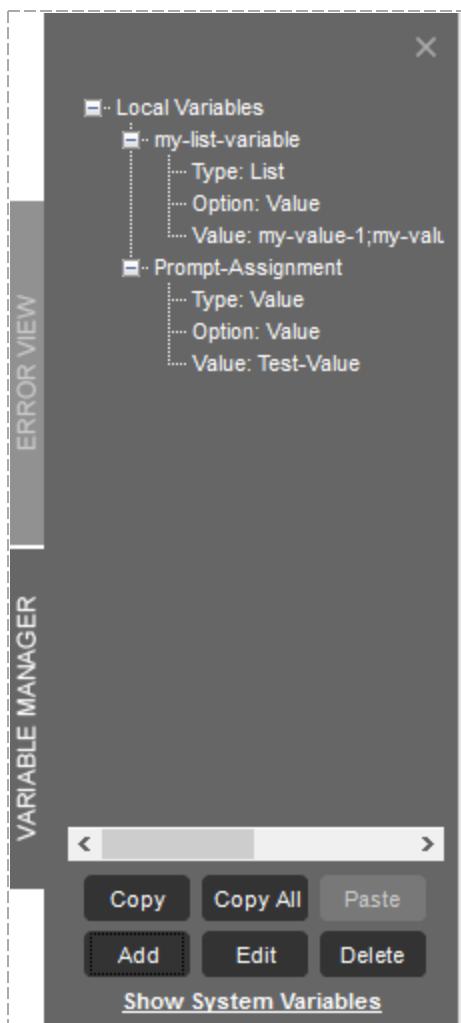Automation Anywhere provides two user variables that are pre-defined for your use:

my-list-variable (type: **List**)

This variable provides a container for a list of values. For more details see List Type Variables

Prompt-Assignment (type: **Value**)

This variable provides a container for a single value. For more details see Value Type Variables .

These variables can be used quickly by pressing the F2 key.

Also See...

- Value Type Variables
- List Type Variables
- Array Type Variables
- Random Variables
- Restoring Deleted Variables

Restoring Deleted Variables

If you delete one or both of the pre-defined user variables (listed below), you can restore them. These include:

- *my-list-variable* (type: List)
- *Prompt-Assignment* (type: Value)

To restore either or both of these variables, follow these steps:

1. Open the Variable Manager.
2. Click on the Add button to display the Add Variable window.
3. Click on the "Select Variable from List" radio button.
4. Check the box next to the pre-defined variable that you want to restore, and click on Save.

> Note: You cannot restore user variables that you create and then delete.

Also See...

- Value Type Variables
- List Type Variables
- Array Type Variables
- Random Variables
- Pre-Defined User Variables

System Variables

Automation Anywhere provides powerful pre-defined system variables that you can use to automate common processes.

System Variable types include:

- *Date/Time:* System-related date and time variables.
- *Loop:* Useful in conjunction with Loop commands.
- Excel: Useful for Excel automation
- *Email:* Useful for Email automation.#
- *Trigger:* Useful in conjunction with triggers.
- *PDF:* Useful for use with PDF command.*
- *System:* Variables specific to a particular client machine.

* Available in Enterprise edition only.
# Available in Enterprise and Small Business editions only.

After system variables are used in a task, you can view their progress while running a task. For more information, see Watching Variables.

Also See...

- Knowing When to Use Variables
- Using the Variable Manager
- Creating a New User Variable
- Assigning Variables to a Task
- Reinitializing Variables

Date and Time System Variables

You can use the set of Date and Time system variables to insert or monitor the current date and time of a system as your automation task runs.

Use Case Scenario

Using Date/Time variables can help you track and log points in time to monitor the success or error of a running process.

To view the system variables and their descriptions, follow these steps:

1. In the Task Editor, click on the Variable Manager icon at the top or on the tab on the right side.
2. Click on the "Show System Variables" link.

To insert a Date/Time system variable, follow these steps:

1. In the Task Editor, drag and drop a Variable Operation command in the task pane.
2. Select the System Variables radio button.
3. In the drop-down menu, select the system variable that you want to insert.
4. Click Save.

Manage your Date/Time variables by using the options listed in the table. The table shows Date/Time system variable names, return values, and actions.

For the Date variable, you can use the System Variables window to change the format that is returned in the Date variable. Follow these steps:

1. In the Task Editor, click on the Variable Manager icon at the top or on the tab on the right side.
2. Click on the "Show System Variables" link.
3. Select the Date variable from the Date/Time list of variables and click the ... button.
4. Select the date format in the Select Date Format pop-up window.

Examples:

A. The following command stores the current DATE, DAY, and YEAR path in a log file. This is important while logging of errors or certain important information on files to include the date stamp.

    1. Drag and drop the Log To File command.

    2. Use the F2 function key to insert variables for $Date$, Day and $Year$.

Command: `Log to File: Date:$Date$Day:$Day$Year:$Year$ in "C:\Variable Operation.txt"`

B. The following command stores the current HOUR, MINUTE, and SECONDS in a log file. This adds the timestamp to the log for analysis.

    2. Drag and drop the Log To File command.

    3. Use the F2 function key to insert variables for $Hour$, $Minute$, and $Seconds$.

Command: `Log to File: Hour:$Hour$Minute:$Minute$Seconds:$Second$ in "C:\Variable Operation.txt"`

Also See...

- System Variables
- User (Local) Variables
- Log to File Command

System Variables - Loop

A Loop system variable complements the Loop command and works within the scope of a defined loop. Most loop variables have no value when used outside of loops. You can manage your Loop command using the variable options in the following table.

Example 1:

```
Start Loop "Each File In Folder D:\My Desktop\JAVA"

   Comment: Please enter your commands to loop. Use $filename$.$extension$ variable for each file
   name in the Loop.

End Loop

Start Loop "Each row in an Internet Explorer Table of Session: Default"

   Comment: Please enter your commands to loop. Use $TableColumn$ variable for each row in Table.

 End Loop
```

Example 2:

1. Comment: ------------------------------------- Start XML Session-------------------------------

2. XML : Start XML Session using existing file "D:\My Desktop\XML Files\Bookstore - 1.xm-l"Session:Session1

3. Comment: -------Get Multiple Nodes, I have set Xpath Xpression as Book [1], means only First Occurance -------

4. XML : Get Multiple Nodes text value from XPath expression "//book[1]/author". Session:Session1

5. Comment: -------------- Use Loop for "Each Node in XML Dataset" -----------------------

6. Start Loop "Each Node In a XML Dataset Session: Session1"

7. Comment: --------- Pring Node Value to Message box using SYSTEM Variable "XML Data Node (Node Name)" ---------

8. Message Box: "$XML Data Node(author)$"

9. End Loop

10. Comment: ------------ Get Multiple Nodes, I have set Xpath Xpression as "Lang", means only All Occurance --------

11. XML : Get Multiple Nodes attributes name "lang" from XPath expression "//book/title ". Session:Session1

12. Comment: -------------- Use Loop for "Each Node in XML Dataset" ----------------------

13. Start Loop "Each Node In a XML Dataset Session: Session1"

14. Comment: ---------Pring Node Value to Message box using SYSTEM Variable "XML Data Node (Node Name)" --------

15. Message Box: "$XML Data Node(lang)$"

16. End Loop

Also See...

- Using Loop Commands
- System Variables
- User (local) Variables

Excel System Variables

You can use Excel System Variables to manage Excel automation processes. These variables are specific to the scope of the Excel command.

Excel relevant system variables

1. **Excel Cell –** This will indicate the column name and position of the cell from the header, when 'Contains Header' is enabled for a session in Open Spreadsheet of the Excel Command. It is applicable to Excel operations like Get Cells, Set Cells, Go to Cell, Delete Cells and Find/Replace.

2. **Excel Cell Column –** This will return the column of the Excel on which the operation has been performed.

3. **Excel Cell Row –** This will return the cell of the Excel on which the operation has been performed.

Also See...

- Excel Command
- Excel Command - Contains Header
- System Variables
- User (local) Variables

Email System Variables

You can use Email system variables within an Email automation task to manage your email automation loop. These variables are specific to the scope of the loop that processes each email message for the server.

Manage your email automation using the e-mail variable options in the table below.

Note: Applicable only for the Loop command: Each Email Message on Mail Server.

When you use the Loop/Email automation command (see example below), a comment in the task explains how you can use these system-defined variables.

Example 1:

1. Start Loop "Each message on server: mail.automationanywhere.com,User Name: support@automationanywhere.com, Server-Type: IMAP, Message Format: HTML"

2. Comment: Please enter your commands to loop. Use Email Automation variables for each email on Server (e.g. $Email From$, $Email To$, etc.)

3. End Loop

Example 2:

In below example we extract emails from inbox and store them in text files. This type of task can be scheduled and emails can be retrieved offline.

1. Start Loop "5" Times

2. Comment: Please enter your commands to loop.

3. Start Loop "Each message on server: imap.gmail.com, User Name: admin1, SSL, ServerType: IMAP, Message Format: HTML"

4. Comment: Please enter your commands to loop. use sys variables for each email on Server.(e.g. $Email From$, $Email To$, etc.)

5. Log to File: $Email From$ John in "D:\Desktop\1.txt"

6. Message Box: "$Email Subject$ $Email From$"

7. Variable Operation: Reset: $Email From$

8. Message Box: "$Email Subject$"

9. Variable Operation: Reset: $Email Subject$

10. End Loop

11. End Loop

The variables in this example can be reset using the Variable Operation (System Variable) command. The drop-down menu offers all of the options for resetting.

Also See...

- Using the Email Automation command
- System Variables
- User (Local) Variables

Trigger Variables

You can use a trigger system variable anytime you work with an automation task that is launched with a trigger. The trigger value is determined based on the type of trigger event.

Using trigger variables is particularly helpful when multiple trigger events are used with an automation task. The variable provides information about the name of the file, folder, window, or service name for the trigger event.

For other uses, this system variable does not contain a value.

The following table provides the return values and description for the trigger variable.
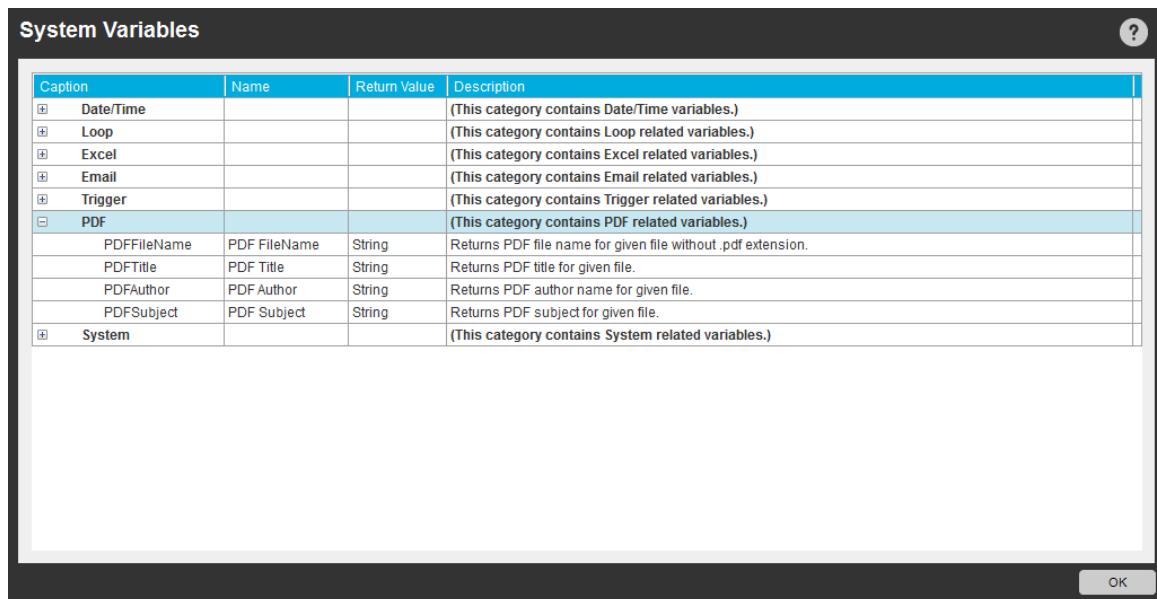
Name	Return Value	Description
Trigger Value	String	Returns the name of File/Folder/Window/Service for which a trigger is fired, depending on the trigger type that was executed.

Also See...

- Using Triggers
- System Variables
- User (Local) Variables

PDF System Variables

You can use the PDF system variables with the PDF Integration command.

The following table provides the names, return values, and descriptions for the PDF system variables.

Name	Return Value	Description
PDFFileName	String	Returns PDF file name for given file without .pdf extension.
PDFTitle	String	Returns PDF title for given file.
PDFAuthor	String	Returns PDF author name for given file.
PDFSubject	String	Returns PDF subject for given file.

Example:

PDF Integration:

Convert "D:\PDF\fw4.pdf" to images as "$PDFTitle$" under folder "D:\PDFImages\"

Also See...

- System Variables
- User (Local) Variables
- PDF Integration Command

System Variables - Specific to System Settings/Parameters

You can use system-related system variables to include parameters in your automation task that are related to a particular computer. The variables return actual system settings and parameters, such as RAM, CPU/RAM usage, and total RAM.

Common Use Case: These variables are useful when the performance of a system needs to be tracked during an activity; for instance load testing.

The following table provides names, return values, and descriptions for the system-related system variables.
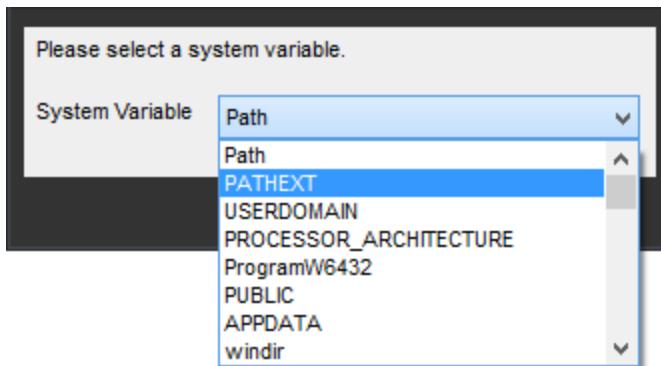
*When you select the System variable, a menu is displayed from which you can select the specific system variable (see steps below).

Steps to select System from Variables:

1. Click F2 and you will see Insert Variable window.

2. Select System and click Insert, a pop up window for System Variable Option appears.

3. Click OK and insert the system variable.

Troubleshooting an error:

Error: If you use the *$CPUUsage$* variable in a message box and an error message is issued, you need to reset the counter for the system by performing the following action:

Open a command prompt and enter the following two commands:

```
cd\windows\system32 [enter]

lodctr /R [enter]
```

Examples

When creating an executable file (EXE) for a task, you might want to use the path of the EXE to log a file or refer to a file located in a folder. In this case, use the *AAApplicationPath* variable to select the path of the EXE from any location in which it is stored. This eliminates the need to use static paths for EXEs.

1. The following command stores the current application path in a log file. Drag and drop a Log To File command to your task, and press the F2 key to insert an *$AAApplicationPath$* variable:

```
Log to File:Automation Anywhere Application Path:$AAApplicationPath$ in "C:\Variable Oper-
ation.txt"
```

2. The following command stores the current RAM usage and CPU usage in a log file. Drag and drop a Log To File command to your task, and press the F2 key to insert *$RAMUsage$* and *$CPUUsage$* variables.

```
Log to File: CPU Usage:$CPUUsage$RAM Usage:$RAMUsage$ in "C:\Variable Operation.txt"
```

Also See...

- Log to File Command
- System Variables
- User (Local) Variables

Working with Variables

These topics describe how you can create and modify variables, as well as take other actions on variables with your automation tasks.

In This Section

- Using the Variable Manager
- Creating a New User Variable
- Adding Variables to a Task
- Editing a Variable
- Copying a Variable
- Deleting a Variable
- Variable Operation: Assign
- Variable Operation: Reinitialize
- Resetting System Variables
- Reading Variables from an External File
- Watching Variables
- Using Variables to Create Timestamps for Your Files
- Passing a Variable to a VB Script
- Workflow Variables through Run Task
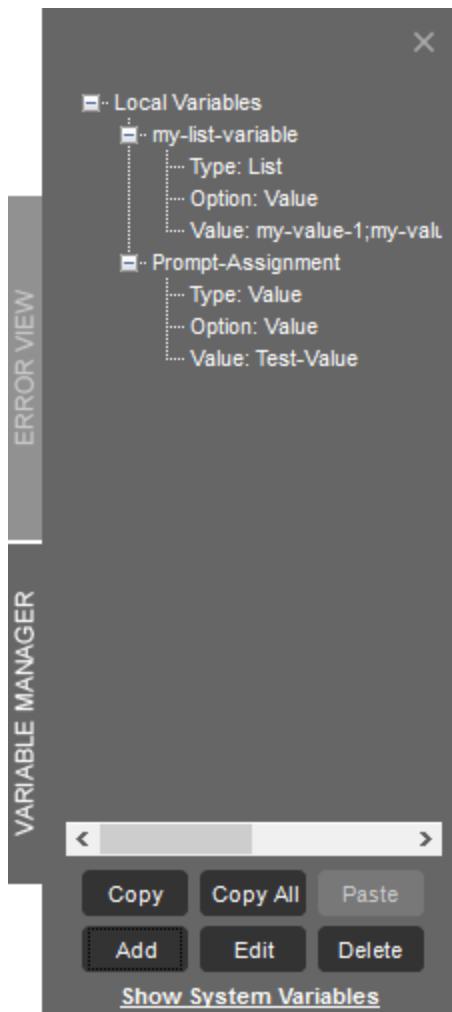
Using the Variable Manager

The Variable Manager displays the local variables that are defined for a particular task. You can use the Variable Manager to add, modify, and copy variables within automation tasks.

In the Task Editor, click the Variable Manager link* on the top menu, or click the Variable Manager tab on the side.

Available in version 8.2 or less

Use the Variable Manager to perform any of the following actions with variables:

- Copy
- Copy All
- Paste
- Add a new variable
- Edit a variable
- Delete a variable

When you click on the "Show System Variables" link, the Variable Manager displays details about all system variables that are available for each task.

When you use the Variable Manager for the first time, you will see that Automation Anywhere provides two pre-defined user variables for your use:

my-list-variable (type: List)

This variable provides a container for a list of values.

Prompt-Assignment (type: Value)

This variable provides a container for a single value.

Also See...

- Types of Variables
- Creating a New User Variable
- Assigning Variables to a Task
- Reinitializing Variables
- Resetting System Variables
- Reading Variables from an External File
- Watching Variables

Creating a New Local Variable

Creating a new local variable and inserting it within your automation task is easy.
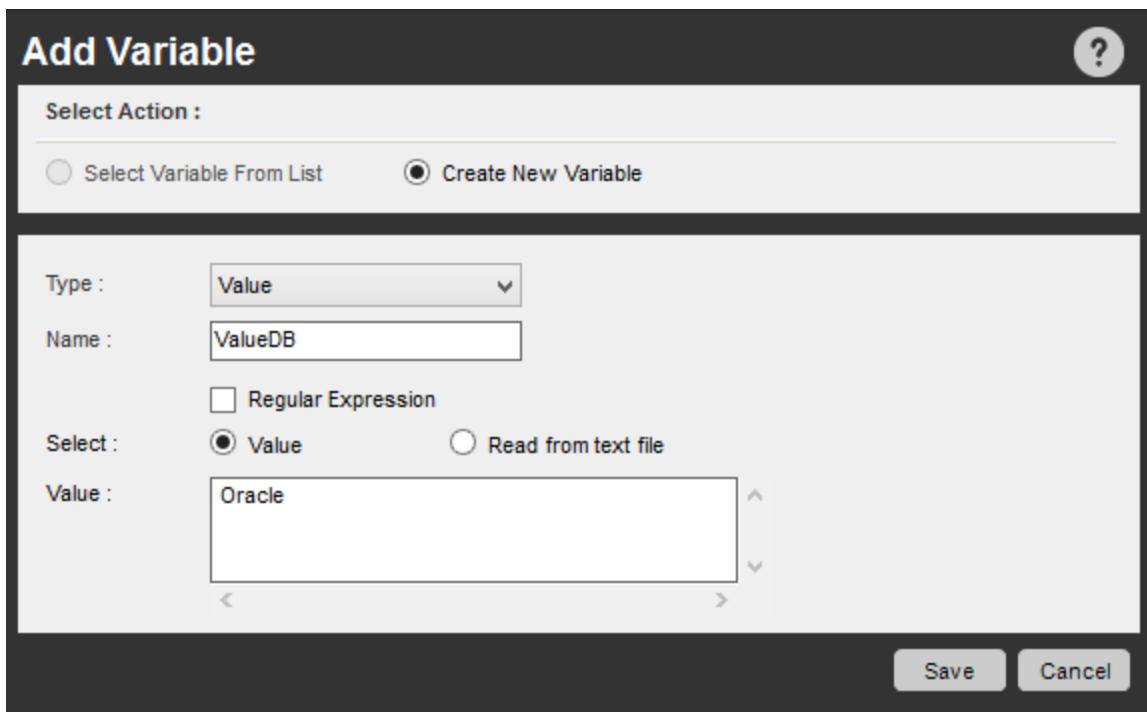
To create a new user variable, follow these steps:

1. In the Variable Manager, click the Add button at bottom or right-click on an existing variable and select Add.

2. In the Add Variable window, the Create New Variable option is selected by default.

3. Select the variable type, enter a name, and select the method for determining the value of the variable. See more details.

Add Variable   ?

Select Action :

◯ Select Variable From List   ⦿ Create New Variable

Type : | Value ▾ |
Name : | ValueDB |

☐ Regular Expression

Select :   ⦿ Value   ◯ Read from text file
Value : | Oracle |

Save   Cancel

Several important facts about using variables include:

- The **$** signs that surround the variable name indicate that the text inside should be treated as a variable.
- When the task runs, Automation Anywhere replaces the variable name with the actual value of the variable.
- All command fields with the light bulb icon support variables. When you want to insert a variable, use the F2 function key to list all user and system variables that are available for selection.
- Variable values can be verified in debug mode by using the Variable Watch feature, which displays the run-time values for the selected variables.

After creating a variable, you can use it within your tasks using most Automation Anywhere commands. Below example illustrates how to insert a variable that will set a cell value in Excel command.

Also See...

- Using the Variable Manager
- Editing a Variable
- Copying a Variable
- Deleting a Variable
- Assigning Variables to a Task
- Reinitializing Variables
- Resetting System Variables
- Reading Variables from an External File
- Watching Variables

Editing a Variable

You can edit and modify any local variable that you have created. In addition, you can edit the pre-defined variables.

To edit a variable, follow these steps:

1. Select the variable you want to edit.
2. Click the Edit button or right-click on the variable and select Edit. The Edit Variable window is displayed.
3. Modify the variable fields as necessary. You can change the variable type, the name, or the method of determining value.
4. Click Save.

Also See...

- Using the Variable Manager
- Creating a New User Variable
- Copying a Variable
- Deleting a Variable
- Assigning Variables to a Task
- Reinitializing Variables
- Resetting System Variables
- Reading Variables from an External File
- Watching Variables

Copying a Variable
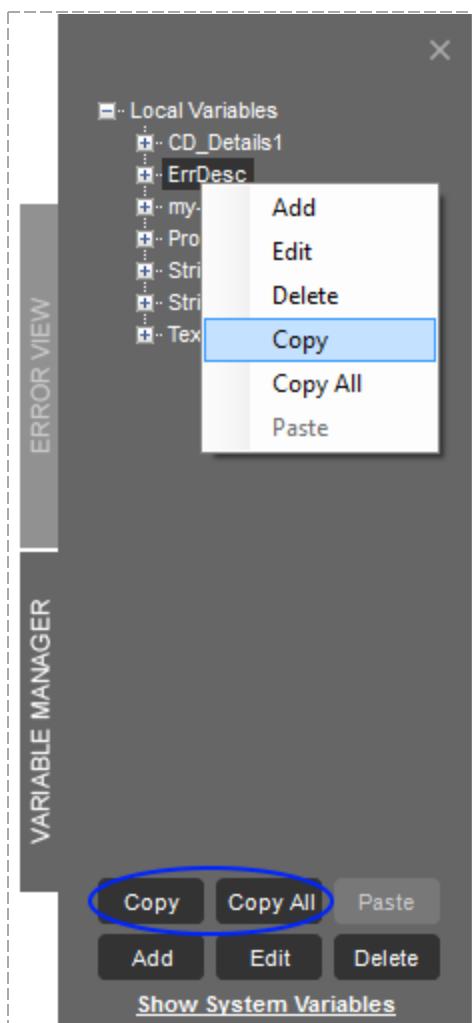
When you need to use a variable from one task in another task, use Copy/Copy All, which are provided in the Variable Manager.

When you select Copy All, you are asked to confirm that you want to overwrite the default variables *Prompt-Assignment* and *my-list-variable*.

To copy a variable, follow these steps:

1. Select the variable you want to copy.
2. Click the Copy button or right-click on the variable and select Copy. You can alternatively click on the Copy All button or right-click and select Copy All. The Copy button is enabled only if a particular variable is selected.
3. Select or open another task in which you want to insert the variables.
4. Click Paste.

The Paste button for a new task is enabled only if variables are first copied.

Also See...

- Using the Variable Manager
- Creating a New User Variable
- Editing a Variable
- Deleting a Variable
- Assigning Variables to a Task

- Reinitializing Variables
- Resetting System Variables
- Reading Variables from an External File
- Watching Variables

Deleting a Variable
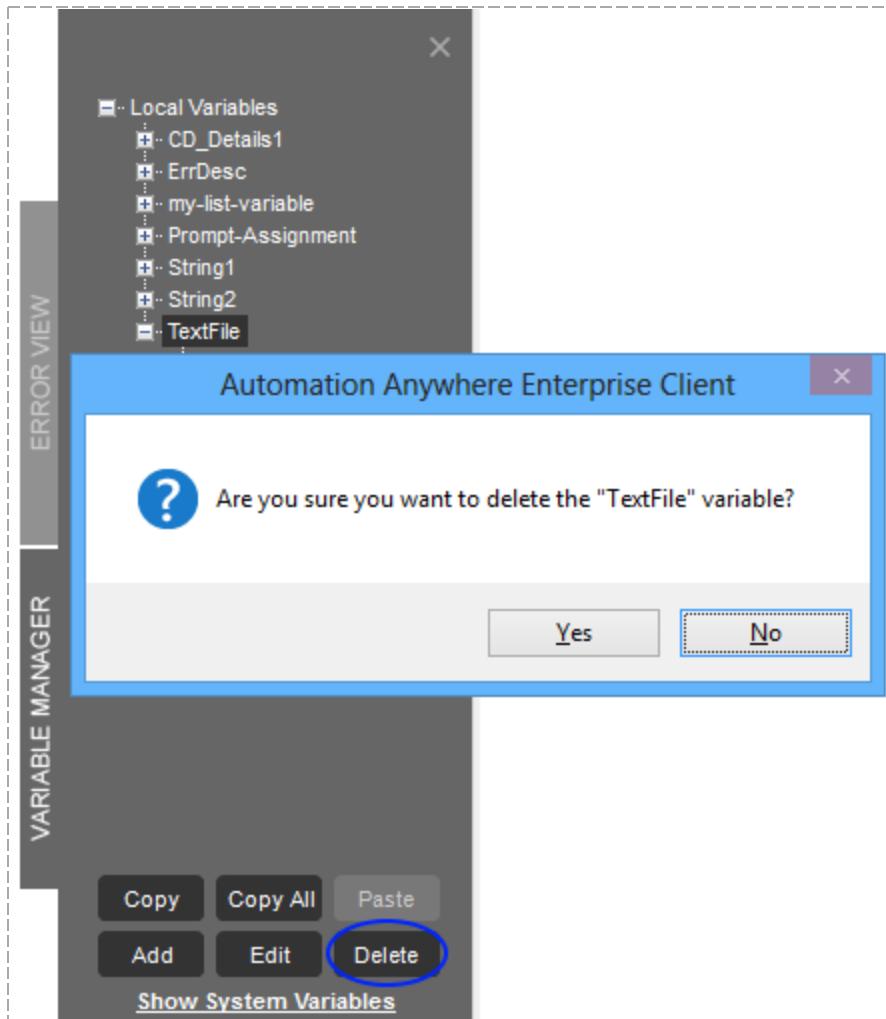
If a variable becomes obsolete and you want to delete it, you can use the Variable Manager to do so.

When using Copy All to copy all variables to another task, you might need to delete several variables that are redundant or useless in the new task.

To delete a variable, follow these steps:

1. Select the variable you want to delete.
2. Click the Delete button or right-click on the variable and select Delete.
3. When the confirmation message is displayed, click yes.

Note: Only single variables can be deleted at one time.

Also See...

- Using the Variable Manager
- Creating a New User Variable
- Editing a Variable
- Copying a Variable
- Assigning Variables to a Task
- Reinitializing Variables

- Resetting System Variables
- Reading Variables from an External File
- Watching Variables

Adding Variables to a Task

Many of the commands that are available in the Task Editor allow you to insert variables in your tasks. Some commands, like Insert Keystrokes, provide a navigation for you to select and insert variables in your task.

For other command fields that allow you to insert variables, use the F2 function key. It will list all user and system variables that are available for insertion.

Also See...

- Assigning Variables to a Task
- Creating a New User Variable
- Editing a Variable
- Watching Variables

Variable Operation: Assign

After you define variables, you can perform operations on them within your automation tasks. Variable Operation command enables you to assign and reinitialize user variables. *Assignment* means that a single value is attached to a variable. This operation enables the user reuse the defined variables as the task runs.

Assignment can be performed on three types of variables: Value, List, and Array. For lists and arrays, the assignment is applied by specific positions within the list or array.

The following table shows the types and source (subtypes) of the user variables and if it supports the assignment operations on them.

Assigning Value Variables

To assign a value to a Value type variable, follow these steps:
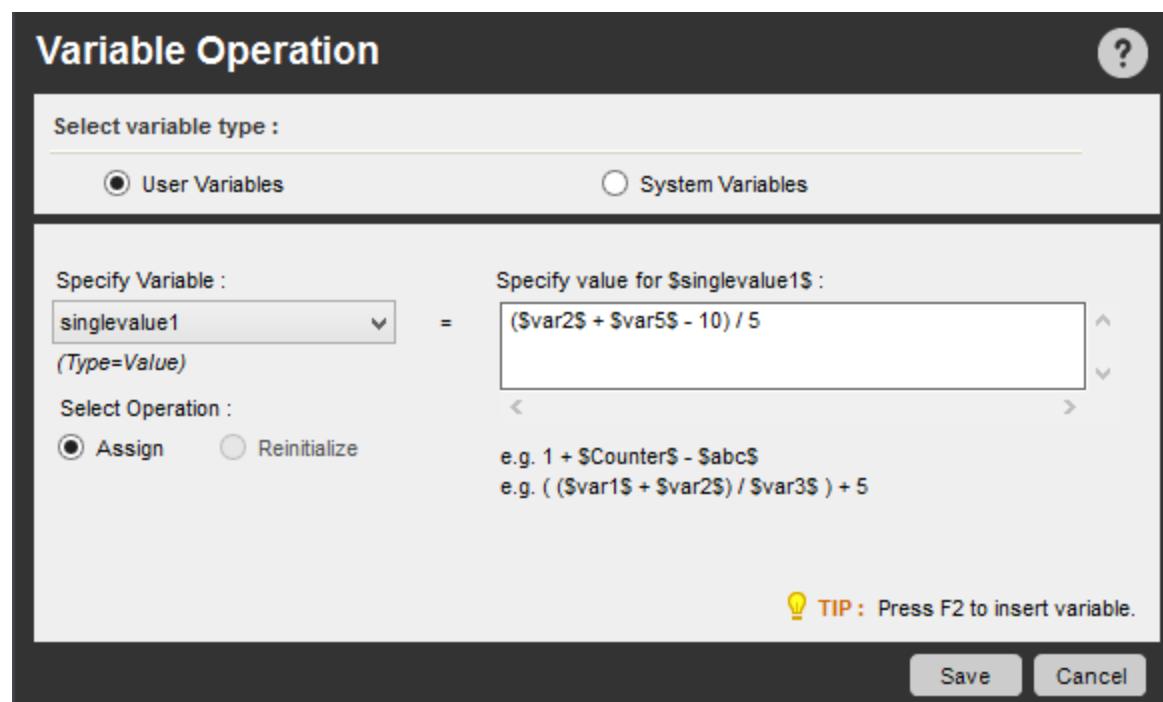
1. In the Task Editor, drag and drop a Variable Operation command into the task.
2. Select the User Variables option.
3. Under "Specify Variable," select the variable from the list of all user variables that are defined for the task. The variable type and source is displayed.
4. Select the Assign operation radio button.
5. In the "Specify value" field, enter a value or insert another variable by pressing the F2 function key.

The right side of the assignment can include any regular operation.

Example: `singleval1 = ($var2$ + $var5$ – 10) / 5`

Operators supported: (,), /, *, +, -

Order of Operations: Parentheses take first precedence in order, with the operation inside them being evaluated from left to right.

Assigning List Variables

When assigning values to a List type variable, you need to specify the position for that value. Any previous value for that position will be overwritten.

To assign a value to a List type variable, follow these steps:

1. In the Task Editor, drag and drop a Variable Operation command into the task.

2. Select the User Variables option.

3. Under "Specify Variable," select the variable from the list of all user variables that are defined for the task. The variable type and source is displayed.

4. Select the Assign operation radio button.

5. For the list variable, the "Select Position" is displayed. Specify the position for this value. For example, `$Counter$`

6. You can also specify another variable to be inserted here by pressing the F2 function key and selecting the variable.

7. In the "Specify value" field, enter a value or insert another variable by pressing the F2 function key.

Variable Operation

Select variable type :

⦿ User Variables   ○ System Variables

Specify Variable :

| email ▾ |

(Type=List)

=

Specify value for $email$:

support@automationanywhere.com

Select Operation :

⦿ Assign   ○ Reinitialize

Select Position :

$Counter$ 💡

e.g. 1 + $Counter$ - abc
e.g. (($var1$ + $var2$) / $var3$) + 5

💡 **TIP :** Press F2 to insert variable.

Save   Cancel

Assigning Array Variables

When assigning values to an Array type variable, you need to specify the positions for the values. Any previous values for these positions will be overwritten.

The source of the array variable can be Excel or CSV files, a database, or a text file, depending on the type of variable you select from the list.

To assign values to an Array type variable, follow these steps:

In the below example we are assigning an array variable for position (2,2) from another variable defined in the task.

1. In the Task Editor, drag and drop a Variable Operation command into the task.

2. Select the User Variables option.

3. Under "Specify Variable," select the variable from the list of all user variables that are defined for the task. The variable type and source is displayed.

4. Select the Assign operation radio button.

5. For the array variable, the "Select Position" fields for Row and Column are displayed. Specify the position for these values. For example, "Row 2 and Column 2".

6. You can also specify other variables to be inserted here by pressing the F2 function key and selecting the variable.

7. In the "Specify value" field, enter values or insert another variable by pressing the F2 function key.

Variable Operation

Select variable type :

( ) User Variables ( ) System Variables

Specify Variable :

arrdb ▼ =

(Type=Array)

Select Operation :

( ) Assign ( ) Reinitialize

Select Position :

Row
2

Column
2

Specify value for $arrdb(2, column)$:

1 + $array1(2,2)$

e.g. 1 + $Counter$ - abc
e.g. (($var1$ + $var2$) / $var3$) + 5

💡 **TIP :** Press F2 to insert variable.

Save   Cancel

Variable Operation: Rounding Values

When using the Variable Operation command in the Task Editor, be aware that values are rounded.

For example, we use a Variable Operation command to assign the value '01.20' to a variable named *Temp*. The returning value in the message box will be *1.2*, as the extra zeros before and after the value are removed.

Examples:

Example 1

Type: List, Source: Read from text example.

1. *Comment: This Loop will execute for 5 times as List read from text file variable has 5 entries.*
2. *Comment: Create Variable List Type : Source = Read from text file having this RDlsttxt = 11, 21,31,41,51*
3. `Start Loop "List Variable $RDlsttxt$"`
4. *Comment: Message box will show the addition of list variable value with other variable value.Here row=10*
5. `Variable Operation: $RDlsttxt$ + $row$ To $Prompt-Assignment$`
6. ` Message Box: "$Prompt-Assignment$"`
7. `End Loop`

Example 2

Type: Array Source: Read from Excel

1. *Comment: We use an array variable to read from existing excel file*
2. `Connect to "$ConnectStr$" Session:'Default'`
3. *Comment: ArrayRows is a system defined variable used to retrieve total rows in the user defined array.*
4. *Comment: ArrayExl - is a user-defined array variable, using a source that is Read from Excel file. In this example, we use a list of books and games having title and cost, which is initialized in the Variable Manager.*
5. `Variable Operation: $ArrayRows($ArrayExl$)$ To $Row$`
6. `Start Loop "$Row$" Times`
7. *Comment: To skip the header title of the csv extracted from the website use below If condition*
8. ` If $Counter$ Equal To (=) "1" Then`
9. `Continue`

10. `End If`

11. `Execute SQL Statement: 'Insert into Ebaykids (BookTitle,cost) values ("$ArrayExl($Counter$,1)$",'$ArrayExl($Counter$,2)$')' Session: 'Default'`

12. `End Loop`

13. `Open "http://www.ebay.com/"`

14. *Comment: Extracting the books on the philosophy into a CSV file using Web Recorder*

15. `Set text 'philosophy books' into _nkw in the webpage 'Electronics, Cars, Fashion, Collectibles, Coupons and More Online Shopping | eBay'`

16. `Click on 'Search' Command Button in the webpage 'Electronics, Cars, Fashion, Collectibles, Coupons and More Online Shopping | eBay'`

17. `Loop While Web Control Exists (Other HTML Control : Next ) in the webpage 'philosophy books | eBay'`

18. ` Extract Multiple Data from Webpage to $filePath$ from the web page: philosophy books | eBay`

19. ` Click on 'Next' Other HTML Control in the webpage 'philosophy books | eBay'`

20. `End Loop`

21. `Extract Multiple Data from Webpage to $filePath$ from the web page: philosophy books | eBay`

Also See...

- Using the Variable Manager
- Creating a New User Variable
- Editing a Variable
- Copying a Variable
- Deleting a Variable
- Reinitializing Variables
- Resetting System Variables
- Reading Variables from an External File
- Watching Variables

Reinitializing Variables

When you create user variables and use them within automation tasks, you need to reinitialize them each time you use them.
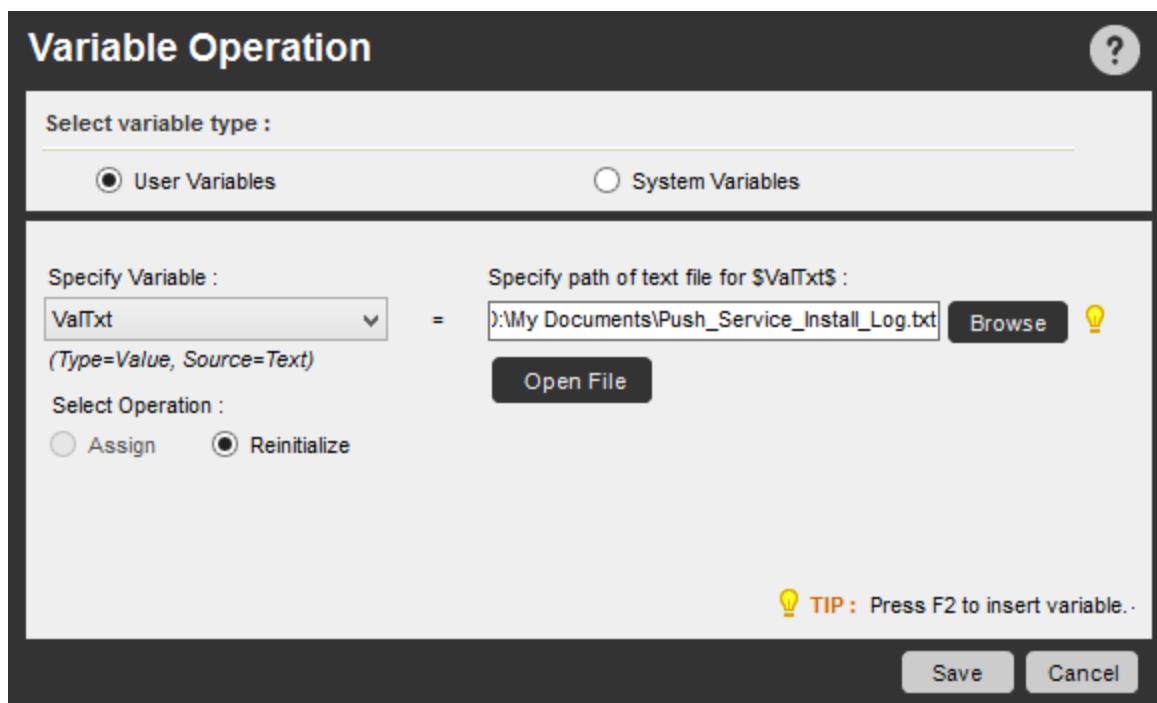
You can reinitialize any type of user variable, including:

- Value variables *(Only read from txt type)*
- List variables
- Array variables
- Random variables

Reinitializing Value Variables

To reinitialize a value type variable, follow these steps:

1. In the Task Editor, drag and drop a Variable Operation command into the task.
2. Select the User Variables option.
3. Under "Specify Variable," select the variable that you want to reinitialize.
4. Select the Reinitialize radio button.
5. In the "Specify value" field, browse to select a new text file for the variable to reinitialize the value.
6. Click Save. Any previous values for this variable will be overwritten.

Reinitializing List Variables

To reinitialize a list type variable, follow these steps:

1. In the Task Editor, drag and drop a Variable Operation command into the task.
2. Select the User Variables option.
3. Under "Specify Variable," select the variable that you want to reinitialize.
4. Select the Reinitialize radio button.
5. In the "Specify value" field, type in new values or browse to locate a new text file.
6. Click Save.

Only the new values that are specified will be assigned. Any previous values for this variable will be overwritten.

Reinitializing Array Variables

Several options are available for reinitializing array variables.

To reinitialize an array type variable, follow these steps:

1. In the Task Editor, drag and drop a Variable Operation command into the task.

2. Select the User Variables option.

3. Under "Specify Variable," select the array variable from the list of user variables that are defined for the task. The variable type and source is displayed.

4. Select the Reinitialize radio button.

5. Specify new dimensions. For example, Row 2 and Column 2. You can also specify other variables to be inserted here by pressing the F2 function key and selecting the variable.

6. In the "Specify value" field, enter values or insert another variable by pressing the F2 function key.

7. Click Save.

Variable Operation

Select variable type :

⦿ User Variables   ◯ System Variables

Specify Variable :

Employee ⌄	=

(Type=Array)

Specify new dimensions for $Employee$:

10 💡 X 5 💡

Row(s)   Column(s)

Select Operation :

◯ Assign   ⦿ Reinitialize

[Initialize Values]

💡 **TIP :** Press F2 to insert variable. ·

[Save] [Cancel]

Restrictions and Usage Rules:

- If the rows and columns are numeric, you can use the Initialize Values button to insert new values. In this case, both the structure and values for the array are completely redefined.

- For an array having source type Excel, you can reinitialize the entire row, column, range, or all of the cells.

- For an array having source type of Excel or CSV file, you need to specify a path in order to reinitialize it.

Variable Operation

Select variable type :

⦿ User Variables   ◯ System Variables

Specify Variable :

Employee_HR ⌄	=

(Type=Array, Source=Excel)

Specify path of excel / csv for $Employee_HR$:

D:\My Documents\Master Output File.csv [Browse] 💡

[Open File]

Select Operation :

◯ Assign   ⦿ Reinitialize

◯ Get All   ⦿ Range

◯ Entire Row   ◯ Entire Column

From : A10 💡 e.g. A1   To : A20 💡 e.g. A5
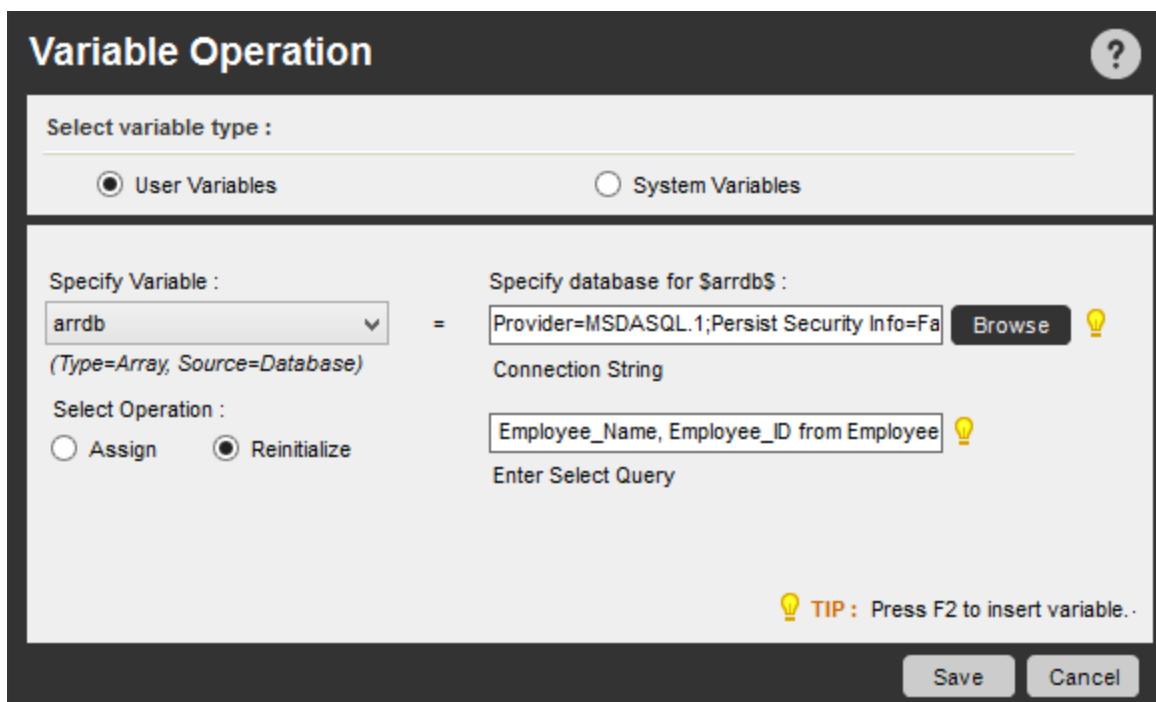
💡 **TIP :** Press F2 to insert variable. ·

[Save] [Cancel]

- The Open File option is enabled only when you specify a valid file path.
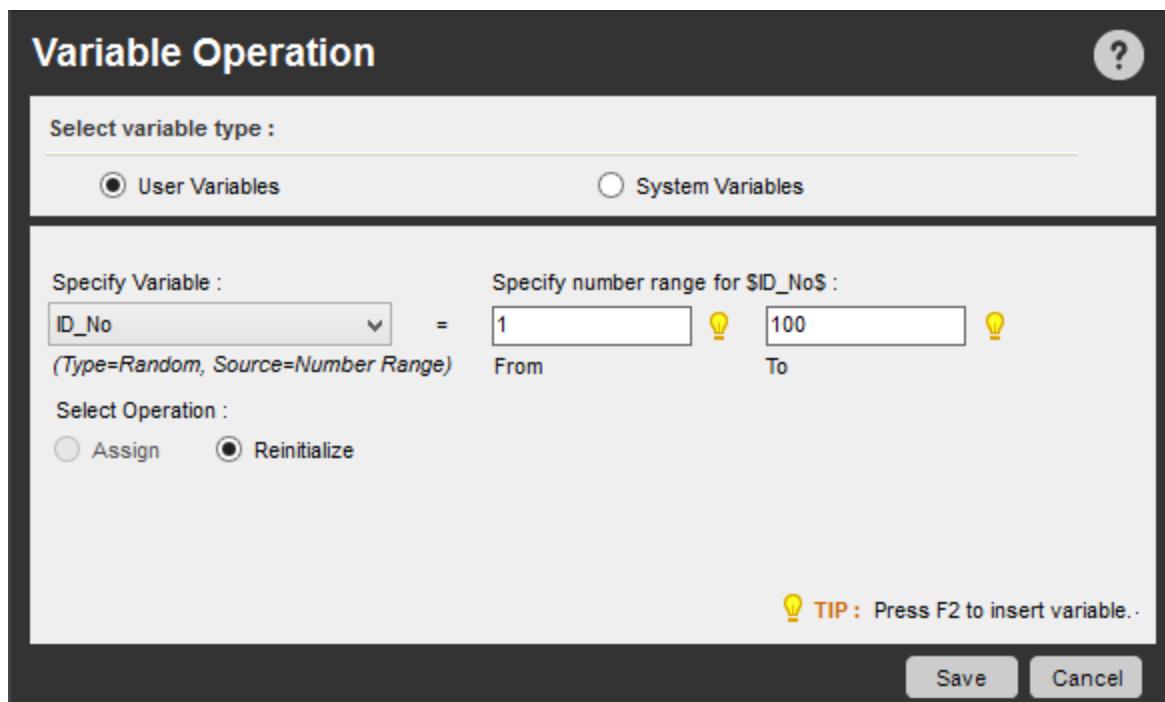
- For an array having a Database source type, you can specify connection properties to a new database with its SQL query.

Reinitializing Random Variables

To reinitialize a random variable, follow these steps:

1. In the Task Editor, drag and drop a Variable Operation command into the task.
2. Select the User Variables option.
3. Under "Specify Variable," select the variable that you want to reinitialize.
4. Select the Reinitialize radio button.
5. In the "Specify value" field, specify new values for the number range or string.
6. Click Save. Any previous values for this variable will be overwritten.

Use Case Scenarios

Example 1:

Type: List Source: Read from text Reinitialize

1. *Comment: Reinitialize Variable Type: List, Source: Read from text file having this RDlsttxt =support@automationanywhere.com, testing@gmail.com, sales@automationanywhere.com, buynow@automationanywhere.com*

2. Variable Operation: Reinitialize $RDlsttxt$

3. Start Loop "List Variable $RDlsttxt$"

4. *Comment: This Loop will run 4 times as 'List read from text file variable has 4 entries.*

5. *Comment: The following command will send emails to the address IDs listed in the above Read from text file -List variable.*

6. Send Email: Subject "Automation Anywhere finished executing the task." with Attachment(s).

7. Message Box: "Email to '$RDlsttxt$' has been sent sucessfully."

8. End Loop

Example 2:

Type: Array Source: Read from Excel

1. *Comment: Reinitializing the Array Source: read from excel file (ArrayExl) with the CSV file extracted from the website.*

2. Variable Operation: Reinitialize $ArrayExl$

3. Variable Operation: $ArrayRows($ArrayExl$)$ To Row

4. Start Loop "Row" Times

5. If $Counter$ Equal To (=) "1" Then

6. *Comment: To skip the header title of the csv extracted from the website*

7. Continue

8. End If

9. Execute SQL Statement: 'Insert into Ebayphilosophy (BookTitle,cost) values ("$ArrayExl($Counter$,1)$",'$ArrayExl($Counter$,2)$')' Session: 'Default'

10. End Loop

Also See...

- Using the Variable Manager
- Creating a New User Variable
- Editing a Variable
- Copying a Variable
- Deleting a Variable
- Assigning Variables to a Task
- Resetting System Variables
- Reading Variables from an External File
- Watching Variables

Resetting System Variables

When using some system variables, you have the option to reset the values while a task is running.
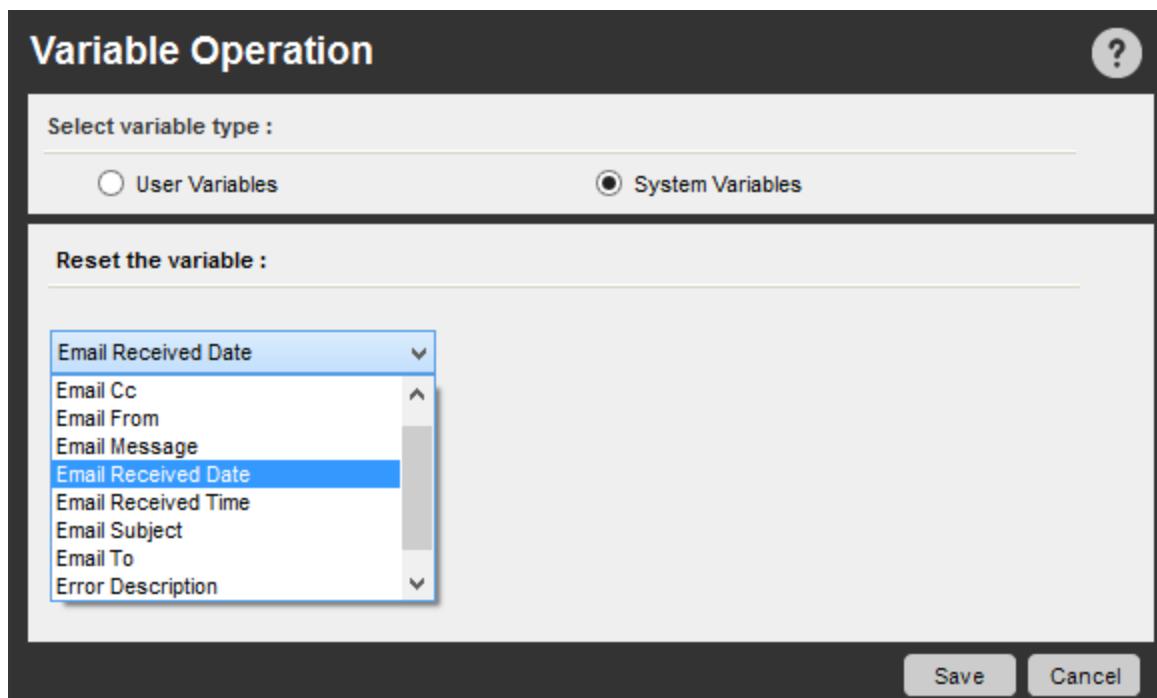
The following system variables can be reset:

- Error Description
- Error Line Number
- Email From
- Email To
- Email Cc
- Email Subject
- Email Message

To reset a system variable, follow these steps:

1. In the Task Editor, drag and drop a Variable Operation command into the task.
2. Select the System Variables radio button.
3. Select the system variable you want to reset.
4. Click Save.

When the task runs, the value for the system variable will be reset.

Also See...

- Using the Variable Manager
- Creating a New User Variable
- Editing a Variable
- Copying a Variable
- Deleting a Variable
- Assigning Variables to a Task

- Reinitializing Variables
- Reading Variables from an External File
- Watching Variables
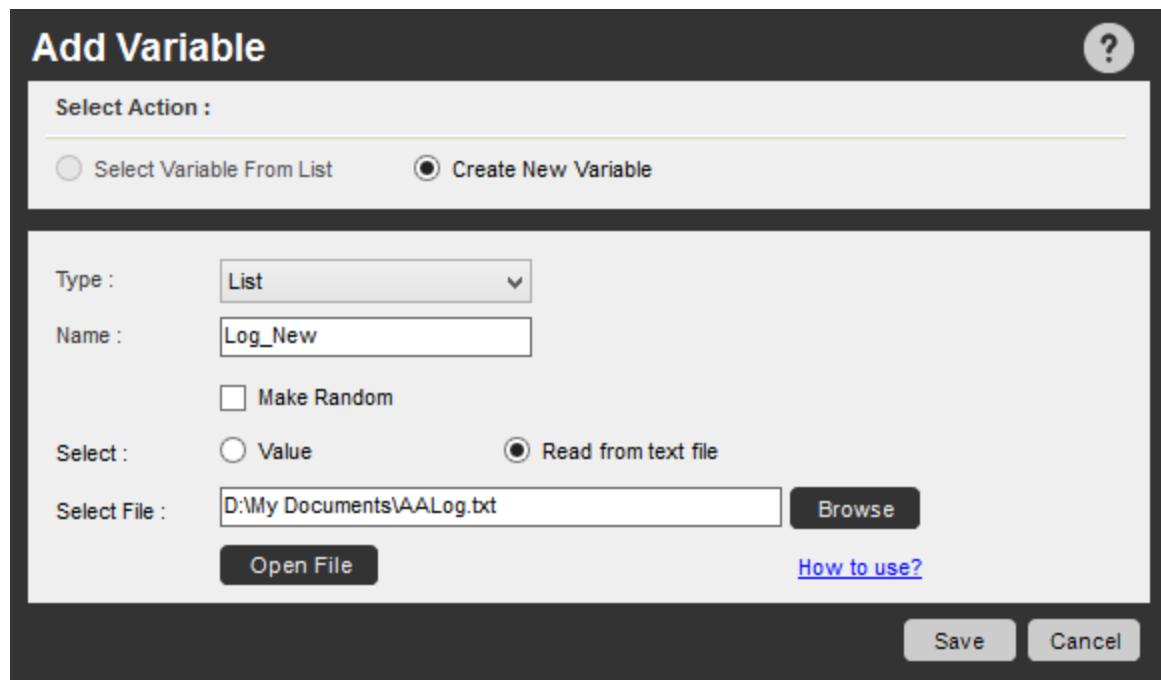
Reading Variables from an External File

Automation Anywhere enables you to set the values of your variables by using an external file. Example files include CSV, Excel, text, and database files.

To read variable values from an external file, follow these steps:

1. In the Variable Manager, click the Add button at bottom or right-click on an existing variable and select Add.
2. In the Add Variable window, the Create New Variable option is selected by default.
3. Select the variable type, enter a name, and select the "Read from Text File" radio button.
4. Specify the following information, depending on the variable type you are creating:
   a. For Value type variables, you can specify a text (.TXT) file that contains the variable values.
   b. For List type variables, you can specify a text (.TXT) files.
   c. For Array type variables, you can specify a text, CSV, Excel, or database file.

Reading from an Excel File

When you create an array variable and specify that values should be read from an Excel file, data can be read for the following:

- For an entire row
- For an entire column
- For a range of cells that you specify using the "From" and "To" fields

Add Variable

Select Action :

○ Select Variable From List   ● Create New Variable

Type :	Array ⌄
Name :	Controls_1
Select :	Read from excel/CSV file ⌄
Select File :	D:\My Documents\Extracted Table.csv **Browse**

Open File   How to use?

○ All Cells   ○ Entire Row   ○ Entire Column   ● Range

From :   A1   e.g. A1   To :   A5   e.g. A5

Save   **Cancel**

Reading from a Database

When you create an array variable and specify that the values should be read from a database file, you need to specify a connection string and a SELECT query.

💡 **Tip:**

To ensure that Automation Anywhere reads text files properly when assigning values for the variables, follow these guidelines:

1. One text file can have multiple *<variable> = <value>* pairs, on separate lines (line returns using the ENTER key).
2. For value type variables, only one value should be assigned.

   Example: *myvar=value1*where *myvar* is the name of the variable.

3. Ensure that your text file is specified in the Variable Manager before the task runs.
4. For List type variables, you can define the comma-separated values for the variable.

   Example: *my-list-var* = value1,value2,value3

5. Variable names are not case-sensitive.

   Example: Text file with different <variable>=<value> pairs:

   ```
   Age=12,13,16,21

   Games=Football,Soccer,Rugby

   username=my-username

   password=my-password
   ```

Also See...

- Using the Variable Manager
- Creating a New User Variable

- Editing a Variable
- Copying a Variable
- Deleting a Variable
- Assigning Variables to a Task
- Reinitializing Variables
- Resetting System Variables
- Watching Variables

Watching Variables

This topic describes how to use the Automation Anywhere Variable Watch feature to watch your variables, when the automated task is running.

To track the progress of variables within the task, Automation Anywhere provides a way for you to watch variables as they change. This makes the progress of variables easier. This can be achieved using "Variable Watch Table" feature which is enabled when you are using the Debugging mode.

The Variable Watch Table displays current variable values while the task runs, and applies to all types of variables (user and system variables) that are defined for that task.

Using the Variable Watch Table

To use the Variable Watch Table, click on the Enable Debugging button at the top of the Task Editor. The Variable Watch Table pops up and displays the actual variable values (or what it contains) as the task runs a particular step.

Initially, the Variable Watch Table is empty. Use the Add button to include variables you want to track.

When the "Add Variables in Watch" window is displayed, select the variables you want to watch and click the Add button.

Any user variables and system variables that are defined for the task can be added to the table.

Add Variable(s) In Watch

Select variable(s) to add in watch :

Variable Name	Return Value	Description
Local Variables		
☐ arrdb	Array	
☐ email	List	
☑ Employee	Array	
☑ Employee_HR	Array	
☐ ID_No	Random	
☐ my-list-variable	List	
☐ Prompt-Assignment	Value	
☐ singlevalue1	Value	
System Variables		
☐ Year	Date/Time	Returns Year. e.g. if date is 02/23/04, it returns 2004.
☐ Month	Date/Time	Returns System Month. e.g. if date is 02/23/04. it returns 2.
☐ Day	Date/Time	Returns System Day. e.g. if date is 02/23/04, it returns 23.
☐ Date	Date/Time	Returns System Date in mm/dd/yyyy hh:mm:ss format.
☐ Hour	Date/Time	Returns System Hour. e.g. if time is 17:33:49, it returns 17.
☐ Minute	Date/Time	Returns System Minute. e.g. if time is 17:33:49, it returns 33.
☐ Second	Date/Time	Returns System Second. e.g. if time is 17:33:49, it returns 49.
☐ Millisecond	Date/Time	Returns System Millisecond. e.g. if time is 17:33:49:10, it returns 10.

☐ Select All

Add   Cancel

As shown below, the Variable Watch Table contains all of the variables selected for watching. The values get populated and changed as the task runs.

Variable(s) Watch Table   X

Name	Value
☐ Employee	
☐ Employee_HR	

☐ Select All   Add   Remove

For an array variable, the Variable Watch Table shows the column/row value. For example, (1,1) is displayed in the left column (Name) in the table.

Variable(s) Watch Table   X

Name	Value
☐ Second	9
☐ Table Column(1)	
☐ TotalRAM	3992
☐ Trigger Value	
☐ XML Data Node(1)	
☐ Year	2014

☐ Select All   Add   Remove

Also See...

- Using the Variable Manager
- Creating a New User Variable
- Editing a Variable
- Copying a Variable
- Deleting a Variable
- Assigning Variables to a Task
- Reinitializing Variables
- Resetting System Variables
- Reading Variables from External Files

Using Variables to Create Timestamps for Your Files

Timestamps are useful tools to use in your automation tasks to track when particular events occur.

The following system variables are available for you to use to create timestamps for your files:

- Year
- Month
- Day
- Hour
- Minute
- Second
- Date

You can combine any of these variables to design your own timestamp formats.

To design date/time timestamps, following these steps:

1. Launch the Variable Manager.
2. Click on "Show System Variables".
3. Select the Date/Time variable and use a combination of variables.

Using the Date Variable with File Names

If you use the Date variable to append a date to file names, you cannot use a slash (/) in your date format. Microsoft Windows prohibits the use of the slash character in file names.

Examples:

1. To create a timestamp for Year, Month, and Day, that is appended to the end of string 'ABC', type: `ABC$Year$$Month$$Day$`

   This example might look like this: *ABC2013521*

2. To create a complete timestamp for the previous example with the current date and time, type: `ABC$Year$$Month$$Day$$Hour$$Minute$$Second$`

   This example might look like this: *ABC201352107:03:03*

3. For clarity, you can insert text characters between the variables.

   For example, to display *ABC2013-5-21*, type: `ABC$Year$-$Month$-$Day$`

Also See...

- Using the Variable Manager
- Creating a New User Variable
- Editing a Variable
- Copying a Variable
- Deleting a Variable
- Assigning Variables to a Task
- Reinitializing Variables
- Resetting System Variables
- Reading Variables from External Files

Passing a Variable to a VB Script

Sometimes, people prefer to run VB (Visual Basic) scripts to accomplish a task or process. In conjunction with Automation Any-where, running VB scripts can be very powerful. These scripts can be automated, using the Run Script command.

Sample Use Cases:

- Obtain data from a file on a website
- Calculate dates and times
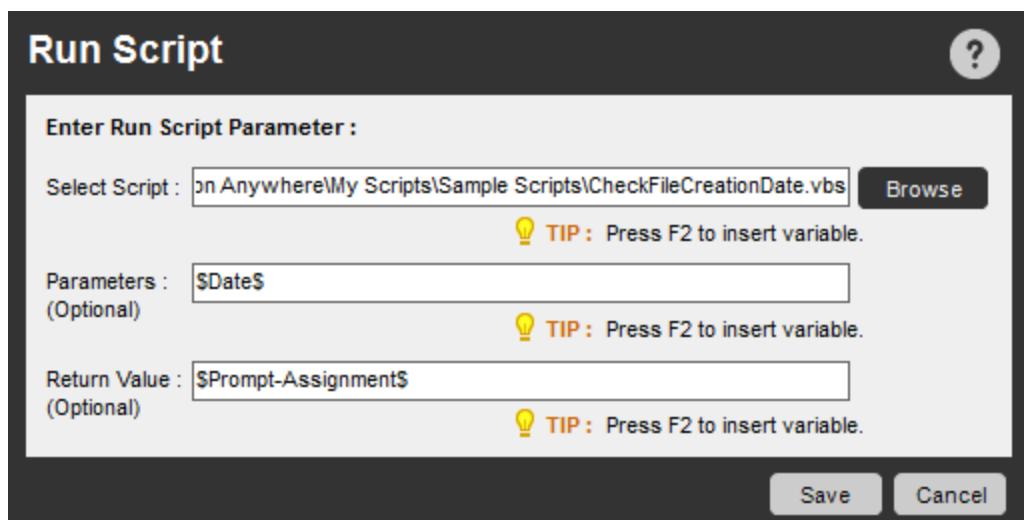
Use the Run Script command to pass values in a variable to the Parameter field. You can then obtain the output from the Return Value field.

Commands that are required to pass values to a VB script and obtain the results include:

1. Pass values in the VB script: WScript.Arguments.Item(0)

2. Return values in VB script: WScript.StdOut.WriteLine "Variable"

For "Variable", the double quotes are not required for Return values.

Separate the values with a space.
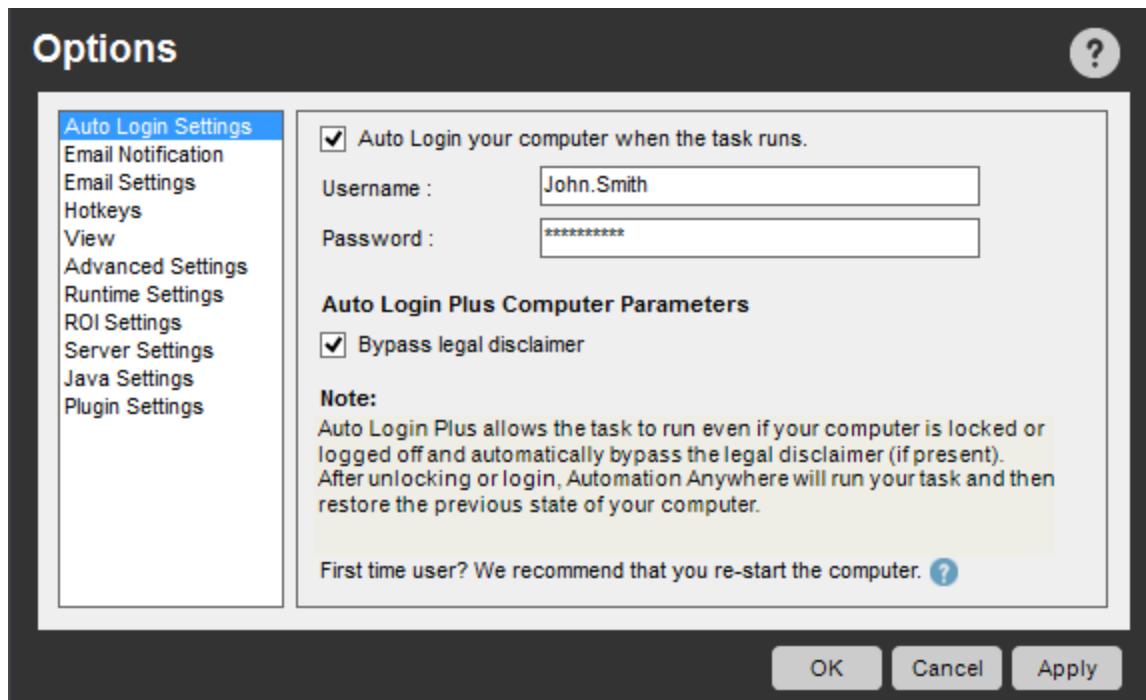
Also See...

- Working with Variables
- Types of Variables

Customizing an Automation Anywhere Client

After installing your Automation Anywhere client, you can customize it by clicking on Tools > Options.

You can customize the following options for your client:

- Auto-Login - to automatically unlock and log into a computer while running an automated task.
- Email Notifications - to notify you about a task's status after it has been executed, by email.
- Hotkeys - to eliminate keystroke and mouse movement errors.
- View Options - to customize the run-time view of an automated task.
- Advanced Settings - to set various options for recording tasks, defining the application locations, editing tasks and enabling system logging.
- Run-time Settings - to customize how your automated process runs and performs, and ensure that the process runs as expected.
- ROI Settings - to calculate the return on investment (ROI) in US dollars.
- Server Settings - to view or set connection properties for accessing and communicating with the Automation AnywhereSmall Business Server.
- Plugin Settings - to install/uninstall plugins for IE 11, Flex and Silverlight.
- Task List View - to customize the columns that you can view in your task list.

Using the Auto Login Option

The Auto-Login option allows you to run tasks on computers that are unattended. Auto-Login unlocks the computer, and runs the automated tasks, and returns the computer to the original state.

You can use in conjunction with the following types of activities:

- Scheduling tasks
- Running reports
- Running workflows
- Triggers

How Auto Login Works

When an automated task runs, Auto Login works as follows:

- Auto-Login checks whether the computer is locked or the user is logged off.
- If the computer is locked, hibernating, or in standby mode, Auto Login unlocks the computer or logs into the machine by using the user name and password provided.
- The automated process runs.
- After the task completes (successfully or with errors), the program restores the computer to its original state.

      Note: Use the Auto Login Advanced Settings to add security features to Auto Login
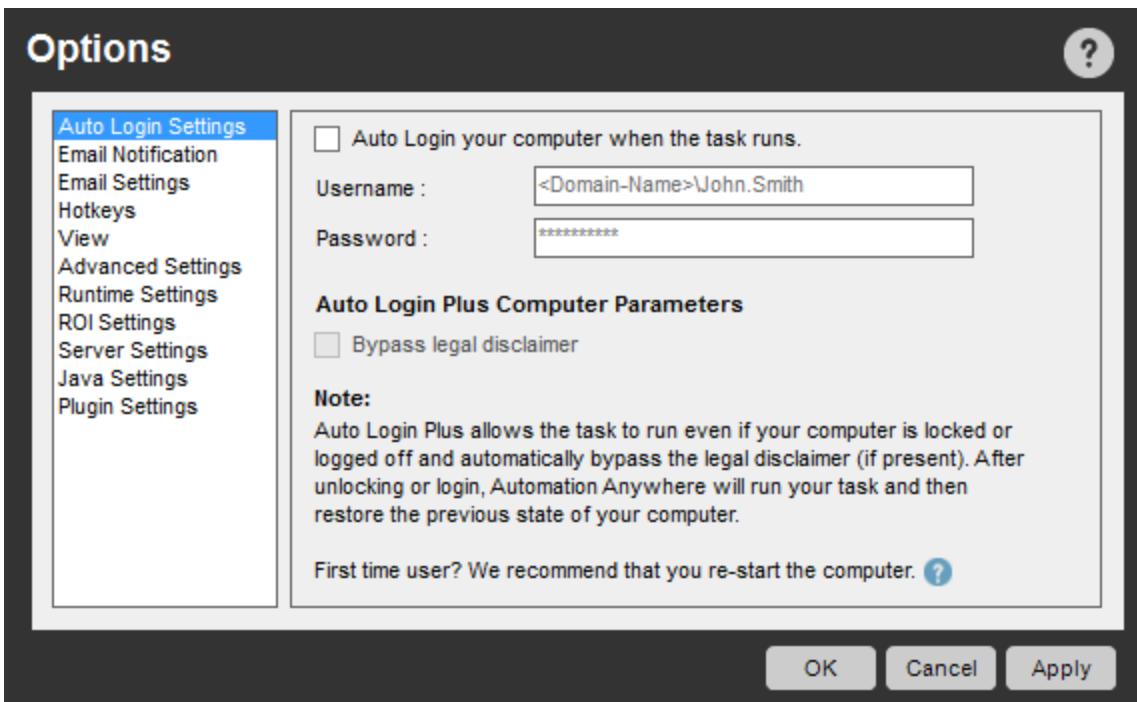
As a task is running, if one of the following actions occur, Automation Anywhere can log off or lock the computer:

- Stop Task key (Escape key by default) on the keyboard is pressed
- Pause key on the keyboard is pressed
- An error occurs while the task runs.

Scheduling Tasks to Use Auto-Login

You can enable the Auto-Login option when you schedule a task to run.

1. Go to Tools > Options > Auto Login.
2. Select the check box for setting Auto Login This shall enable you to apply Auto Login to all of your tasks.

(Image as seen in Enterprise Edition 8.2)

3. Select a task and go to the Schedule tab.

4. Input the required parameters and click save:

5. Input the username and password in the 'Login to Schedule Task' window. Click Save.
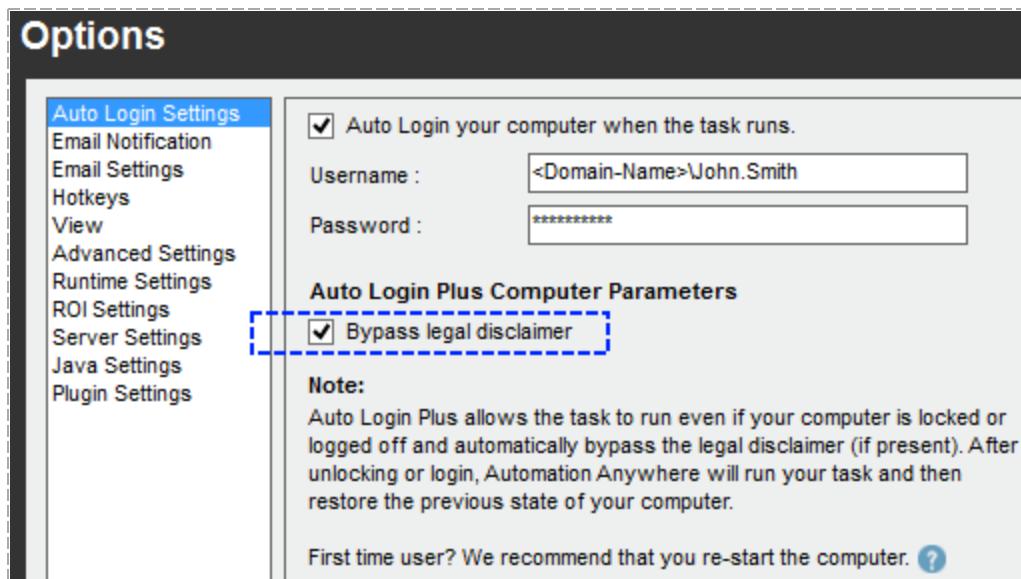
Refer Scheduling a Task for details.

- To turn off Auto-Login, click on the Tools > Options. Under the Auto-Login option, deselect the 'Enable Auto-Login your computer when the task runs' check box.

- If an error occurs while a task is running, Automation Anywhere returns the computer to its previous state, regardless of other tasks waiting to process.
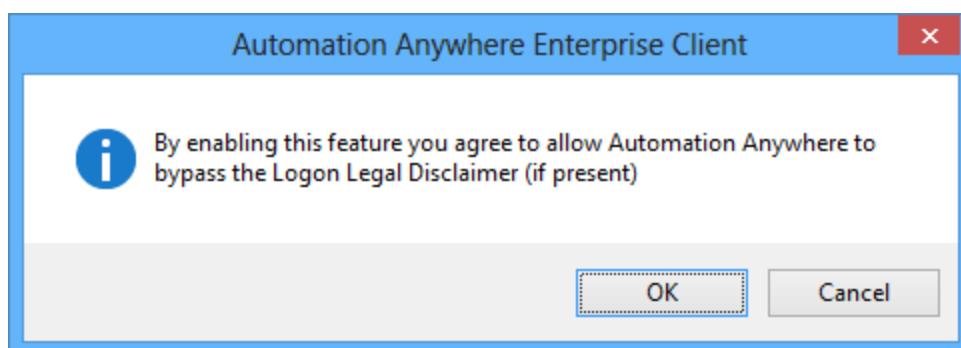
```
Note: Do not use interactive commands (commands that prompt a user for input) for tasks that use
the Auto-Login feature.
```

Enabling Bypass Legal Disclaimer *

* Available in Enterprise Edition 7.5 onward. For details contact support.@automationanywhere.com

If you operate your tasks in an environment that require stringent security measures, you may not be able to log on automatically to run certain tasks. Given this scenario, to make your Auto Login option work, you may be required to enable the 'Bypass Legal Disclaimer' feature.

The feature is available for use, when you select 'Auto Login your computer when the task runs' option. If you wish to bypass the security measures and automatically log on, enable 'Bypass legal disclaimer'.

⚠ `Caution Note: Please note that this feature should be used with utmost care before enabling the same. The sole responsibility of using this feature lies with the user and Automation Anywhere shall not be held liable for its misuse and/or related consequences.`

Operating System Support
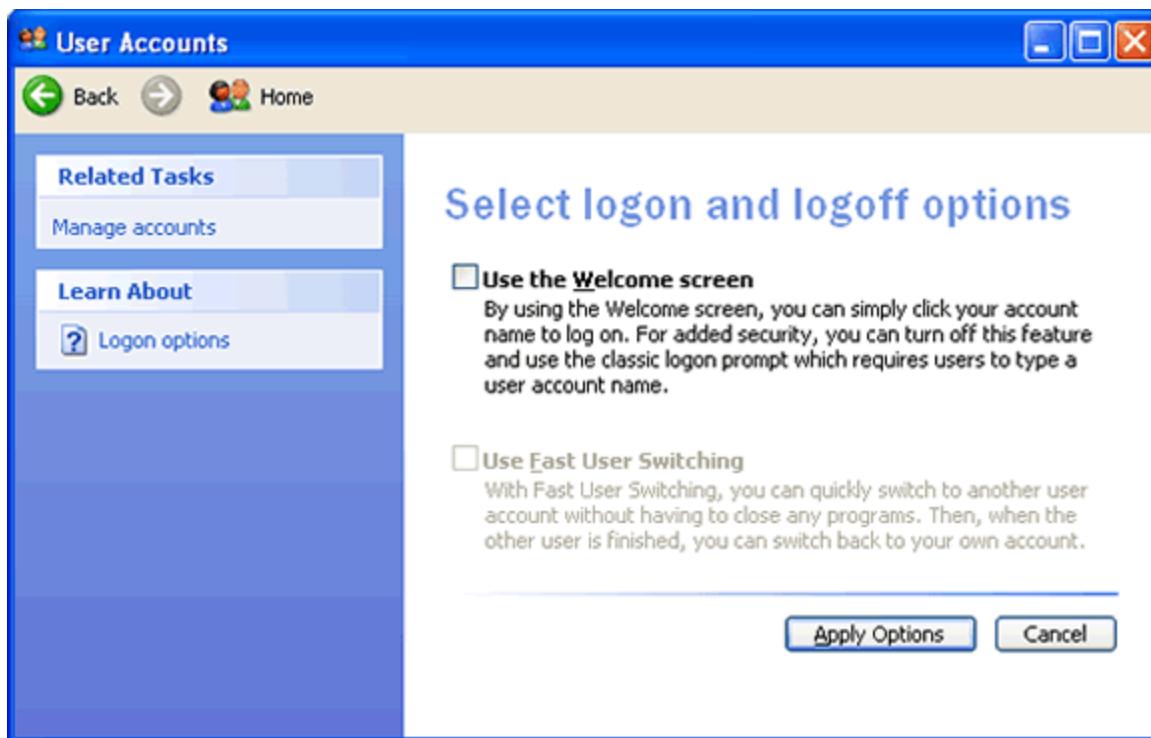
Auto-Login is supported for the following operating systems:

- Windows XP (only for the classic log on screen)
- Windows 2000
- Windows 2003
- Windows Vista
- Windows 7
- Windows 8

Windows XP Users

You are required to disable the 'Welcome' screen and 'Fast User Switching' in order to be able to use Auto-Login:

1. On the Windows desktop, go to Start > Settings > Control Panel.
2. Click User Accounts.
3. Click 'Change the way users log on or off.'
4. Deselect the 'Use the Welcome Screen' check box.
5. Click on Apply Options.

- Use the following screen shot to verify that you are using the correct screen.

Also See...

- Using the Advanced Settings
- Setting up Email Notifications
- Defining Client Hotkeys
- Setting View Options
- Using Run-Time Settings
- Using ROI Settings
- Using Server Settings
- Customizing Task List View

Setting up Email Notifications
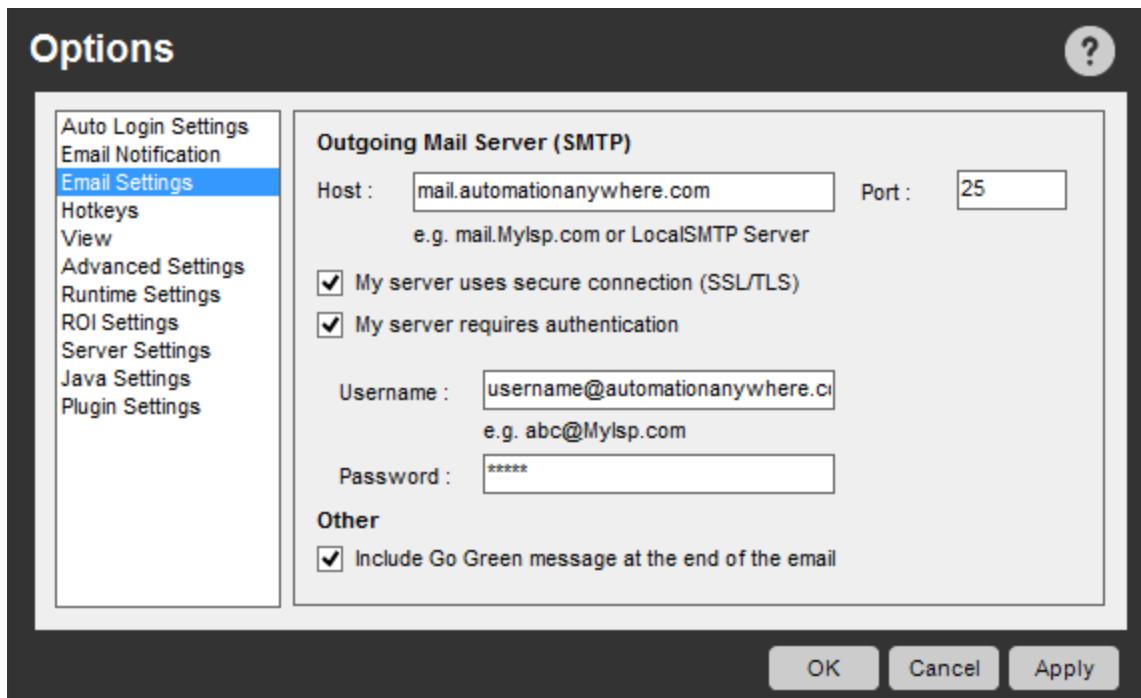
Automation Anywhere client enables you to notify task status after it has been executed, by email.

To enable email notifications, ensure that you configure your email settings first. For this follow the steps given here:

1. In the main Automation Anywhere window, click on Tools > Options.

   > 📝 Note: You can also configure email notification settings when you click on Properties tab > Notification

2. Click on Email Settings

3. Now set the following fields given under Outgoing Mail Server (SMTP):

- **Host -** The SMTP Host enables you to send email from an Automation Anywhere client over the Internet. You can obtain this information from your Internet email provider, or from the configuration section of your email program.

- **Port -** This setting defines the TCP/IP port that Automation Anywhere uses to send email. The default port is 25, the Internet standard for sending SMTP email.

- **Username -** If your outgoing mail server requires a user name and password, specify the user name in this field.

- **Password -** If your outgoing mail server requires a user name and password, specify the password in this field.
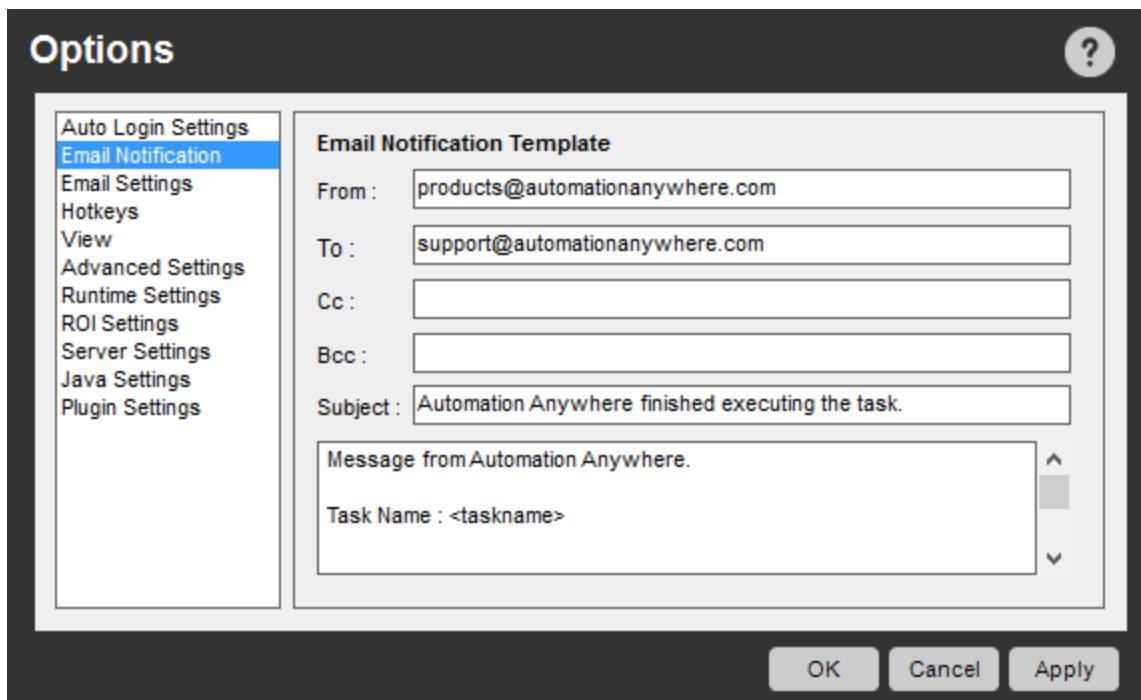
🌱 Go Green !

- **Other-**Automation Anywhere cares about the environment and wants to help spread the message. As part of our Green Campaign, a message is included at the end of each email notification. You can deselect this option if you do not want your outgoing emails to include the Go Green message.

The next step involves setting the email notification options:

1. Click on Email Notification to set notification content. Enter a "From" and "To" email address and one or more email addresses to be notified.

2. Also include a subject line and main body content for the email.

💡 Tip:

1. Always try to include the task name with the variable <taskname>.

2. If you plan to notify only one email address with task updates, use the Tools -> Options -> Email Notification to simply set it one time. However, if you want different tasks to send status emails to different email addresses, specify the email addresses within each task.

Also See...

Defining Client Hotkeys

You can define Hotkeys to save time and eliminate keystroke and mouse errors.

Defining Hotkeys Using Automation Anywhere

You can define hotkeys for the following operations:

- Start recording
- Stop recording
- Stop a running task

These actions have hotkeys defined by default. To select different keys on the keyboard for these actions, simply press the "Hotkey..." button and select a key combination from the list.
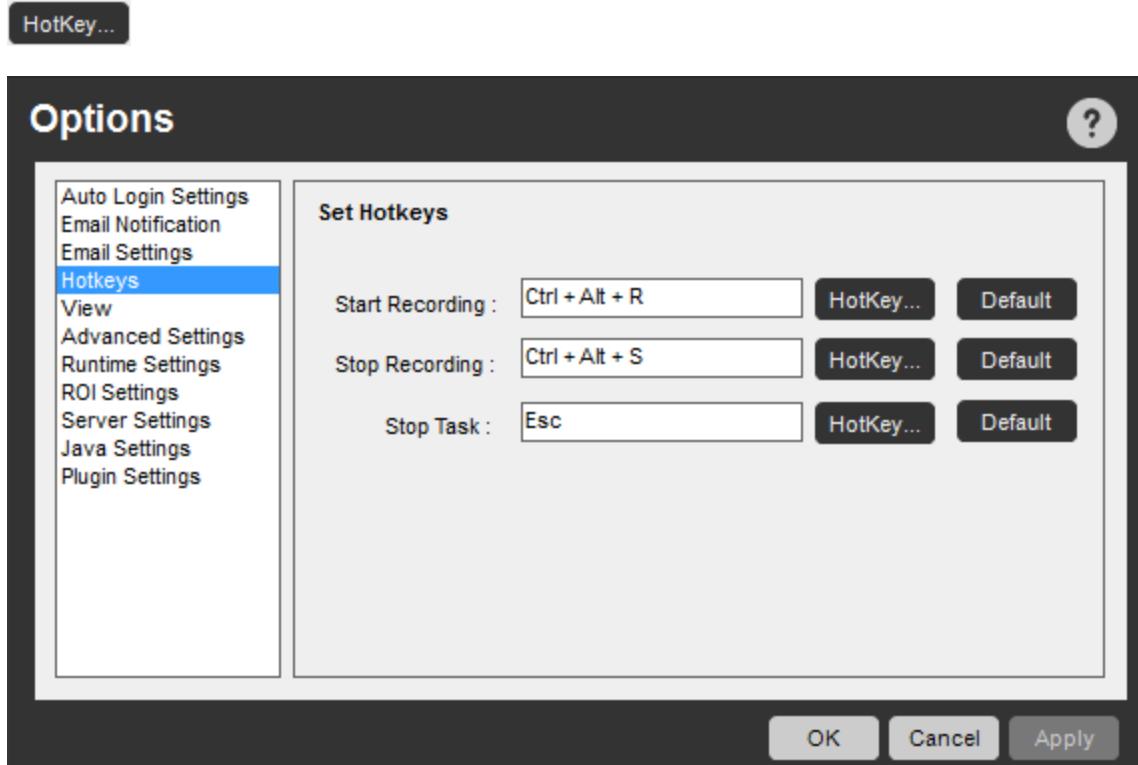
To reset the action to the default, press the "Default" button.

Note: To use hotkeys, ensure that Automation Anywhere is running in the Windows system tray.

Configuring a New Hotkey

Follow these steps to configure a new hotkey:
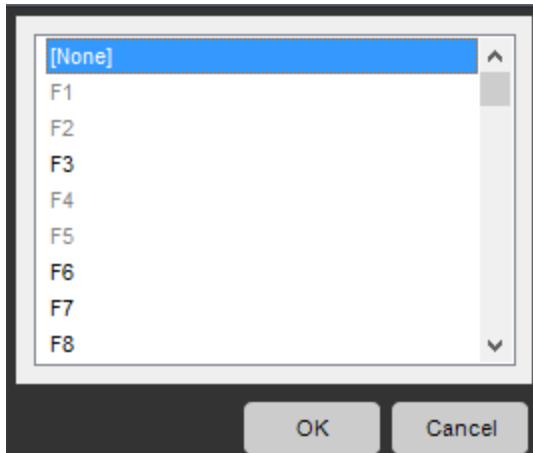
1. From the main Automation Anywhere window, click on Tools -> Options
2. In the Options window, click on Hotkeys
3. Set the Hotkeys for 'Start Recording', 'Stop recording' and 'Stop Task' by clicking the Hotkey button.

Note: You can modify a hotkey by pressing the Hotkey button. Also set the selected key as default.

Alternately, you can select Properties tab -> Hotkey -> [...] to set hotkeys for the selected task

4. Select an action and a key combination from the list.
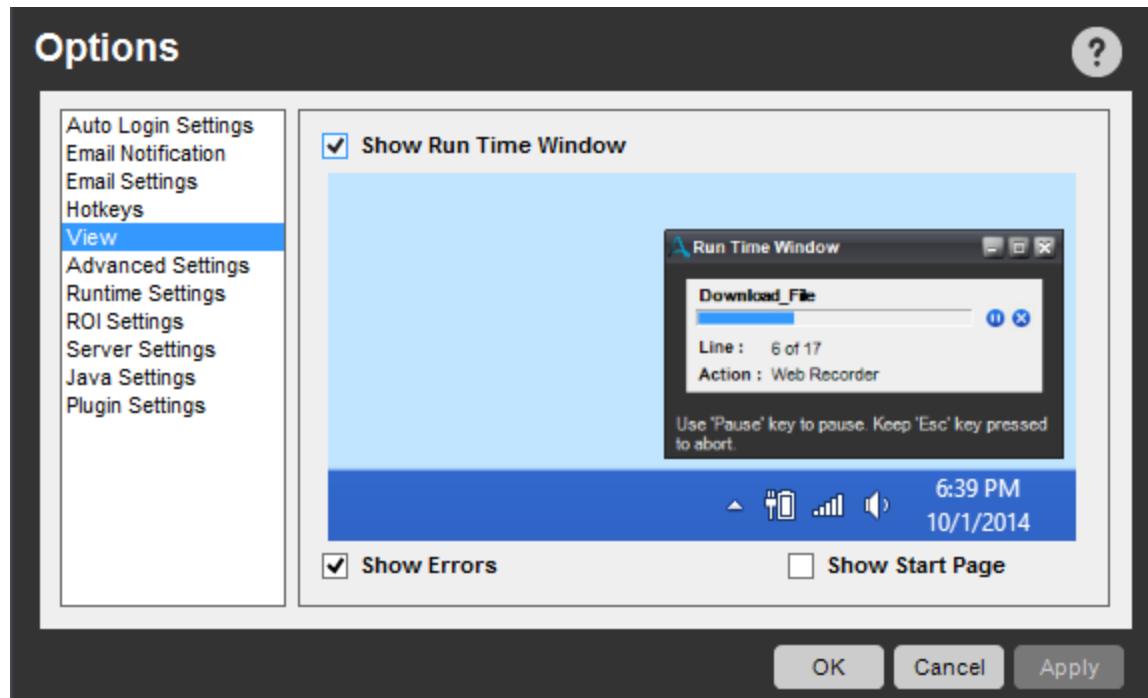
5. Click OK.

Also See...

- Using Auto-Login Option
- Setting up Email Notifications
- Setting View Options
- Using Advanced Settings
- Using Run-Time Settings
- Using ROI Settings
- Using Server Settings
- Customizing the Task List View

Setting View Options

Viewing options provide you the necessary leverage to control the way certain options are displayed. For instance, you can customize to display the run-time window and keep tab upon the error(s) the task encounters when running.

(Image as seen in Enterprise edition 8.0.1)

Configure your options from Tools > Options > View:

1. **Show Run Time Window:** Show or hide the Run Status window that Automation Anywhere displays in the lower right side of your screen while a task runs.
2. **Show Errors:** Show or hide any errors that Automation Anywhere issues while running a task.
3. **Show Start Page:** Show or hide the Automation Anywhere Start Page when starting up the Client.

Also See...

- Using Auto-Login Option
- Setting up Email Notifications
- Defining Client Hotkeys
- Using Advanced Settings
- Using Run-time Settings
- Using ROI Settings
- Using Server Settings
- Customizing Task List View

Using Advanced Settings

Using advanced settings options enables you to customize the way you control mouse moves/clicks, keystrokes, capture screen-shots etc when recording a task.

Configure these settings from Tools > Options > Advanced Settings:

For Recording:

1. **Record Mouse Moves -** To stop recording mouse moves while recording a task, un-check this option. Some mouse moves have application-specific meaning, such as when recording application menus. In these cases, keep this option enabled.

2. **Record Mouse Clicks -** To stop recording mouse clicks while recording a task, un-check this option.

3. **Record on Mouse Down -** Check/select this option to record a task with the help of Web Recorder using mouse down event i.e. while scrolling.

   > Note: This setting is specific for Web Recorder task only.

4. **Record Keystrokes -** To stop recording keystrokes while recording a task, un-check this option.

5. **Capture Screen-shots While Recording -** To capture and display images of the screen-shot while you record tasks use this option. For more information about this feature, see Using SnapPoint.

For Application Location:

1. **Application Path -** Check this to specify a different application path. The default application path is the Automation Anywhere Files folder under My Documents.

   When changing this location, all of your tasks will be saved in the new location. The new path will take effect when you restart the Automation Anywhere client.

   > Note: After changing the application path, all the triggers, hotkeys, and scheduled tasks will run as normal.

For Editing and Logging:

1. **IEdit Task on double click in Task List -** By Default, a double click on a task will run or execute a task. Enable this option to change the default setting to edit a task on a double click.

2. **Enable System Logging -** For more information about this feature, see Viewing System Logs.

For Proxy Server:*

Applicable from version 8.1 onward for Small Business and Enterprise editions only

Proxy Server Settings (For Web Service) - These are applicable for Web Service command only. If your environment is using a proxy server, specify the Host Name/IP Address and the Port Number.

Also See...

- Using the Auto-Login Option
- Setting up Email Notifications
- Defining Client Hotkeys
- Setting View Options
- Using Run-Time Settings
- Using ROI Settings
- Using Server Settings
- Customizing Task List View
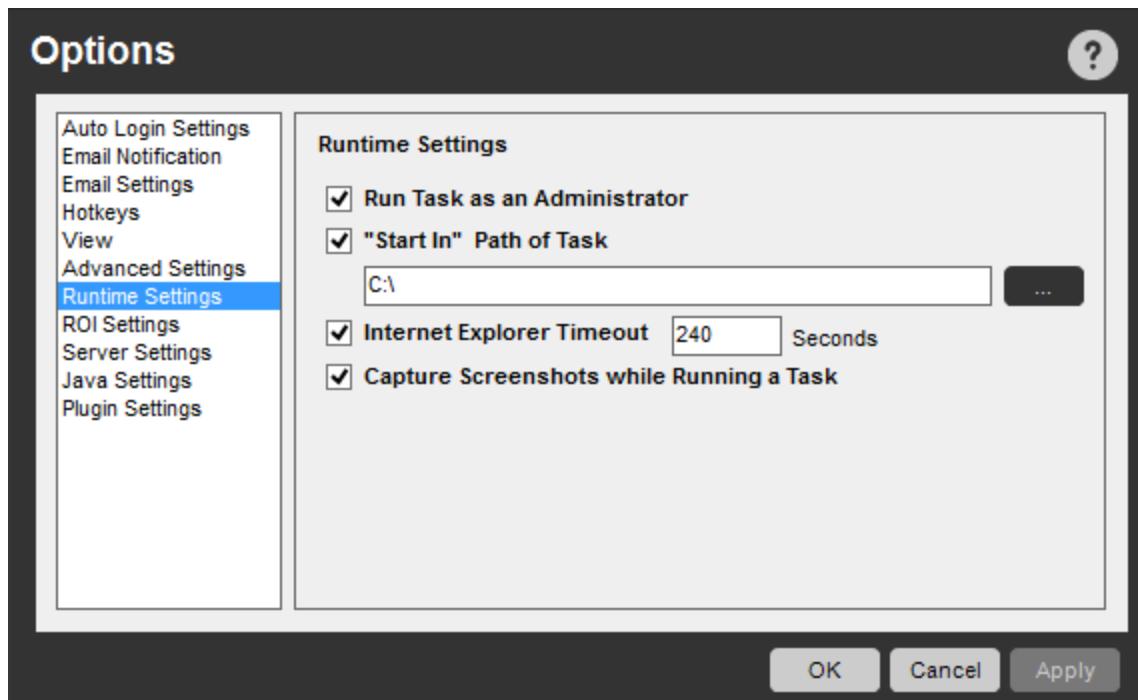
Using Run-Time Settings

The Automation Anywhere run-time settings enable you to customize how your automated task runs and performs, and helps to ensure that it runs as expected.

After recording your task, you should check and adjust these settings. The run-time settings take effect, when you initiate a run of your automated task.

Settings: An Overview

> Note: Applicable to Windows Vista or later, in which user account control (UAC) is enabled. For other operating systems or when UAC is not enabled, turn this setting off.For details refer Setting User Access Control.

Run Task as an Administrator

Use this setting when running critical tasks for which a Windows prompt is issued asking for authorization to run a process. When this setting is turned on, the user name and password of the computer administrator is required to be entered before the task can be run.

"Start In" Path of Task

Use this setting when a specific path is not specified in the Log-To-File command or other commands, in which a file or folder path is required for running a task.

While the task runs, a new folder is created in the location specified in this setting field. After you apply this setting, all newly created files and folders are located in this path location.

Internet Explorer Timeout

Internet connection speeds can differ widely. Hence adjustments are required to ensure proper running of a task which involves loading web pages.

Use this setting to adjust the timeout value, based on your local internet connection speed or the time taken by the web page to load completely.

For example, if you use a slow dial-up connection and experience frequent timeout errors while a task is running, adjust this setting to increase the timeout period.

Capture Screen-shots while Running a Task

Use this setting to capture visual logs (screen-shots) as a task runs. You can turn this setting off by un-checking the check-box.

> 💡`Tip: Turning this setting off can enhance performance and free up hard drive memory.`
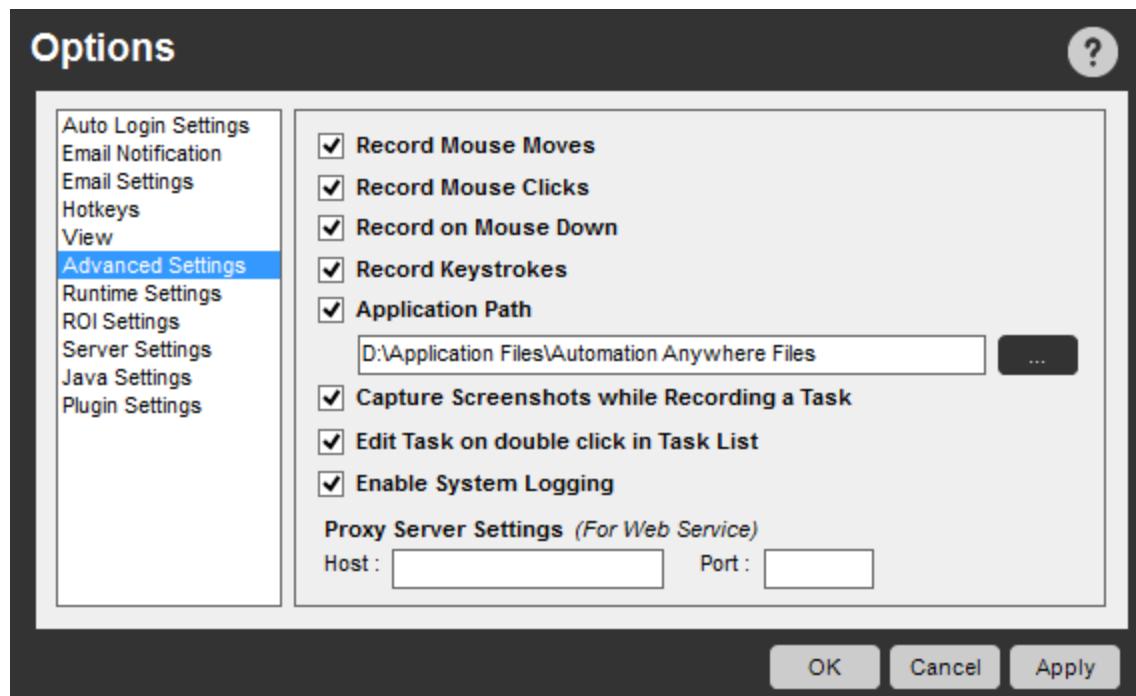
For more information on capturing visual logs, see Using SnapPoint.

Also See...

- Using the Auto-Login Option
- Setting up Email Notifications
- Defining Client Hotkeys
- Using Advanced Settings
- Using ROI Settings
- Setting View Options
- Using Server Settings
- Customizing Task List View
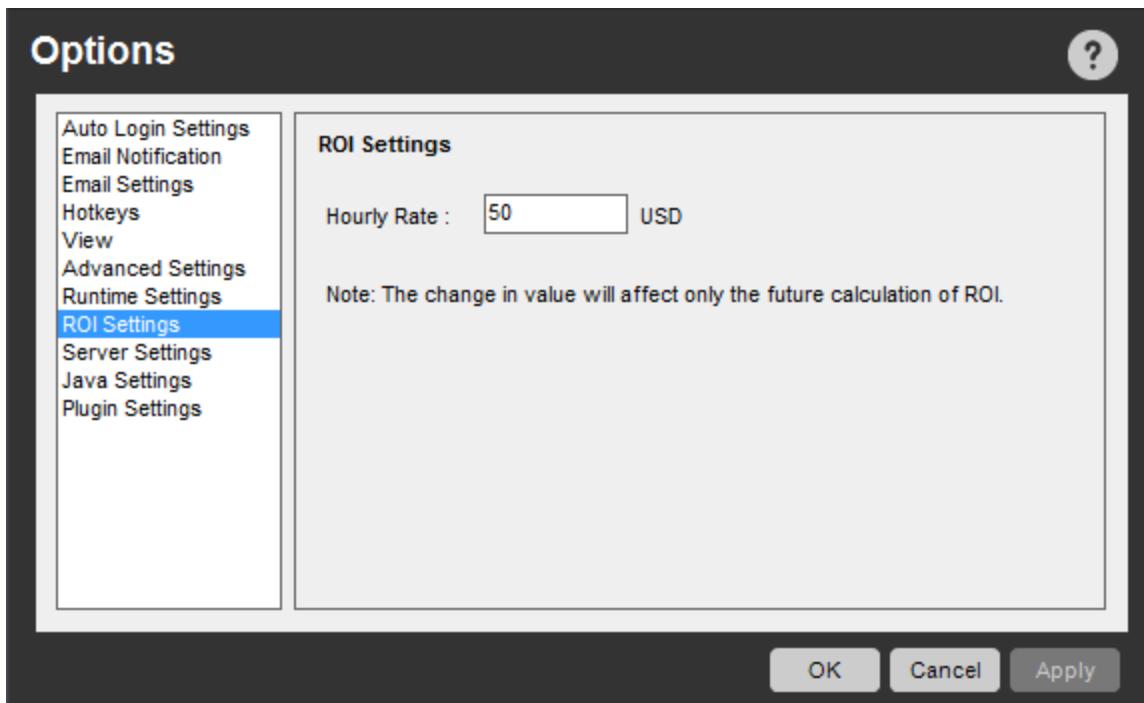
Using ROI Settings

Automation Anywhere calculates the return on investment (ROI) that results from automating tasks.

The ROI is a cumulative savings amount in U.S. dollars that is calculated as you run automated tasks.

> Note: The ROI calculation is based on cost of hourly time, run time, debug time, alert assistance time, documentation time, training time, and knowledge capture.

- Click on Tools -> Options -> ROI Settings to change the hourly rate. Changes to this value will be reflected in future ROI calculations.

Also See...

- Using the Auto-Login Option
- Setting up Email Notifications
- Defining Client Hotkeys
- Using Advanced Settings
- Using Run-Time Settings
- Setting View Options
- Using Server Settings
- Customizing Task List View
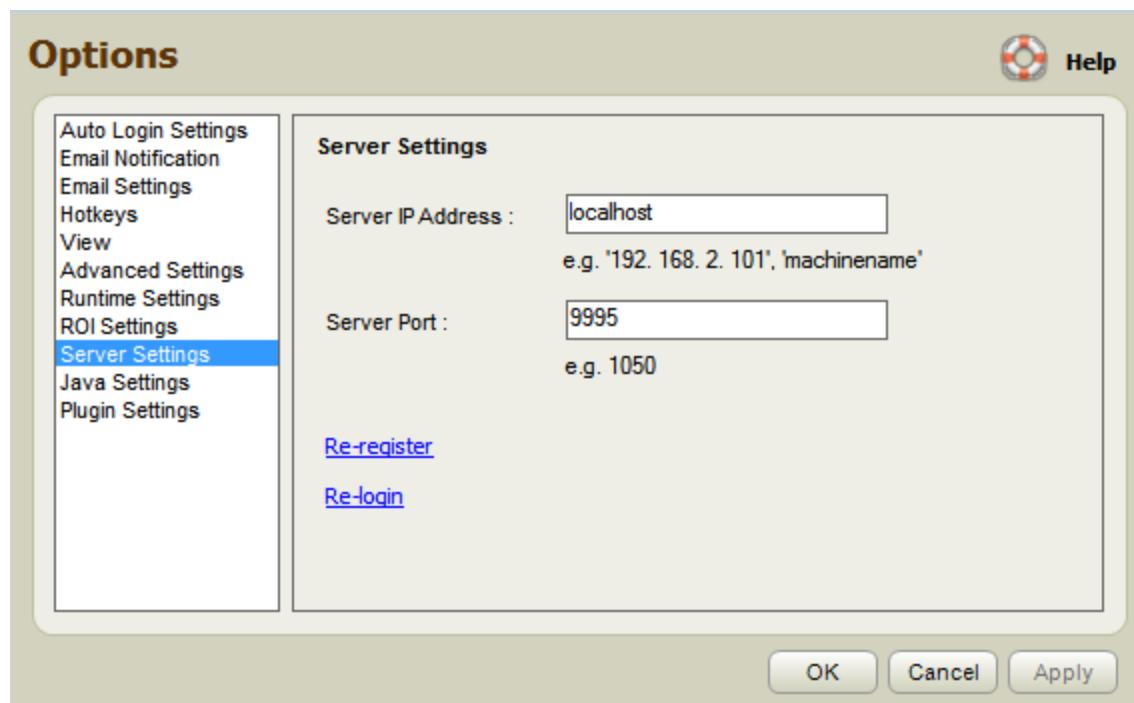
Using Server Settings

Use the server settings to view or set connection properties for accessing and communicating with the Automation Anywhere server.

Configure the Server Settings to:

- Specify a server IP address.
- Specify a server port.
- Re-register a client with the server especially when you want to login as a different client not registered earlier. Refer Registering a Client for details.
- Re-login to the server to login as a different user from the same machine. Refer Logging into the Server for details.
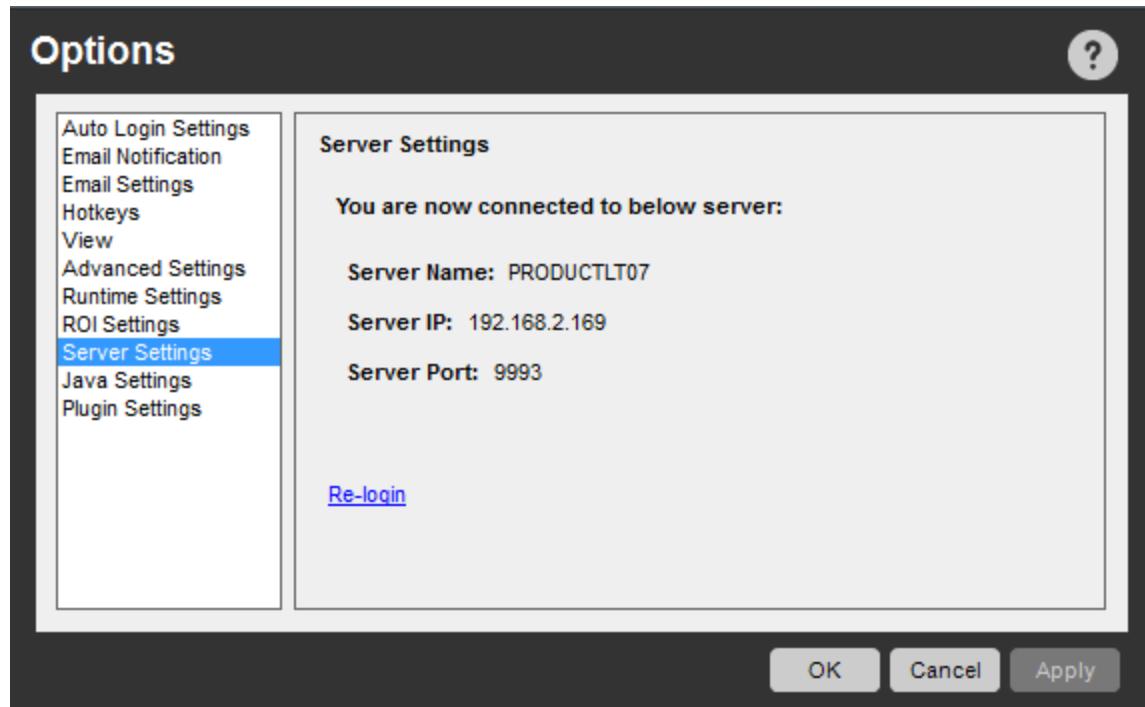
To request access permissions from the server administrator, see Communicating with the Server.

- To configure the server settings, go to Tools -> Options -> Server Settings.
- Specify or edit any properties necessary.

For Small Business and Enterprise editions from version 7.5 onward

The current server settings are displayed as default.

To re-configure and/or login using different credentials, click 'Re-login'.

Also See...
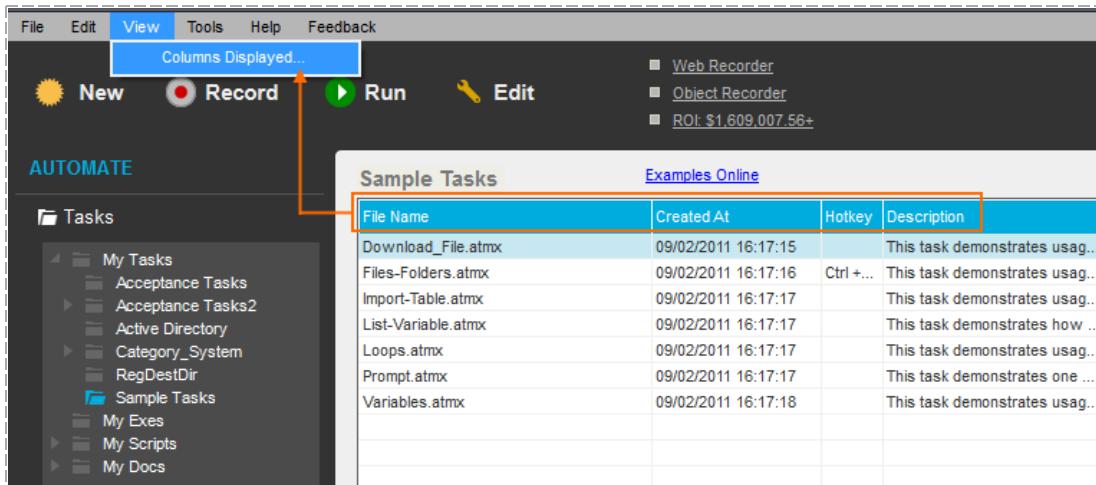
- Using the Auto-Login Option
- Setting up Email Notifications
- Defining Client Hotkeys
- Using Advanced Settings
- Using Run-Time Settings
- Setting View Options
- Using ROI Settings
- Customizing Task List View

Customizing the Task List View

At times, for ease of use you might want to remove certain columns from being displayed in the Task List.
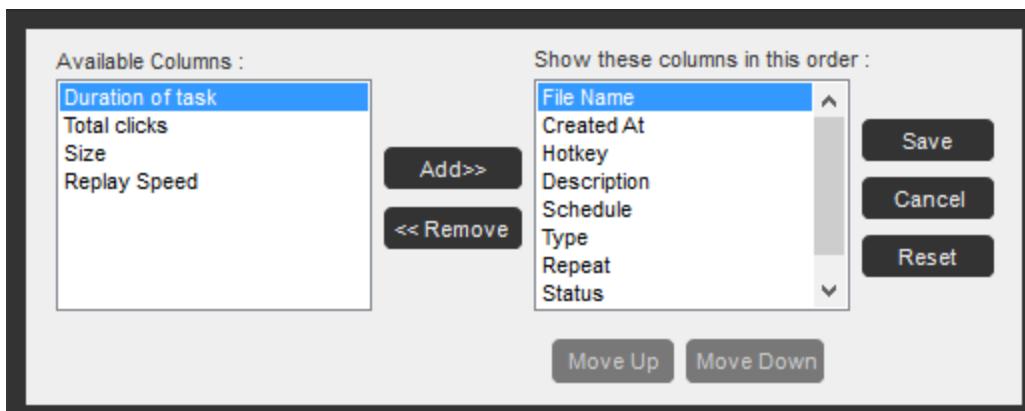
You can customize your view of the Task List columns by following these steps:

1. From the main Automation Anywhere window, click on View -> Columns Displayed.

2. Add, remove, or reorder columns in the display.

3. Save, cancel or reset your changes to the display.

Adding a Column

Select a column from the list of available columns and click Add.

The newly added column is displayed in the list on the right.

Removing a Column

Select a column from the list on the right depicted below the text 'Show these columns in this order:' and click Remove.

The column is removed from the list and is added back to the list on the left side (Available Columns)

> Note: The File Name column is required and cannot be removed from view.

Reorder Columns

Use the Move Up and Move Down buttons to change the left-to-right column position in the view. The columns that are listed from top-to-bottom are displayed in the Task List view from left-to-right.

Save Changes

Click Save to save any changes you make to the view and apply them to the Task List.

Cancel Changes

Click Cancel to abandon any changes you make to the view.

Reset Display to Default

Click Reset to restore the default column settings in the Task List view.

> Note: The default column order is: File Name, Type, Repeat, Status, and Last Run Time.

Also See...

- Using Auto-Login Option
- Setting up Email Notifications
- Setting View Options
- Using Advanced Settings
- Using Run-Time Settings
- Using ROI Settings
- Using Server Settings
- Defining Client Hotkeys

Setting User Access Control and Data Execution Prevention

In some cases, when using Automation Anywhere on Windows Vista, Windows 7 etc User Access Control (UAC) and Data Execution Prevention (DEP) settings need to be modified.

To ensure smooth operation on machines that run on OS Windows Vista and above, follow these steps:

1. Check to see whether User Access Control (UAC) is turned on for the computer that is running Automation Anywhere:

    - On the Windows desktop, click on Start > Control Panel > User Accounts > Change User Account Control Settings.
    - Drag the slider to the bottom so that the setting is on "Never Notify".

2. Add Automation Anywhere to the list of exceptions under Data Execution Protection (DEP):

    - On the Windows desktop, click on Start > Control Panel > System > Advanced System Settings.
    - On the 'Advanced' tab, click the 'Settings' button in the 'Performance' area.
    - Click on the tab "Data Execution Prevention" and select the option "Turn on DEP for all programs and services except those I select:"
    - Click on the 'Add' button, and add the Automation Anywhere.exe (Program Files (x86)\Automation Anywhere) folder to the list.
    - Click on Apply and then OK.

3. Reboot the computer to ensure that the new settings take effect.

Also See...

- Using Auto-Login Option
- Setting up Email Notifications
- Defining Client Hotkeys
- Setting View Options
- Using Advanced Settings
- Using Run-Time Settings
- Using ROI Settings
- Using Server Settings
- Customizing the Task List View
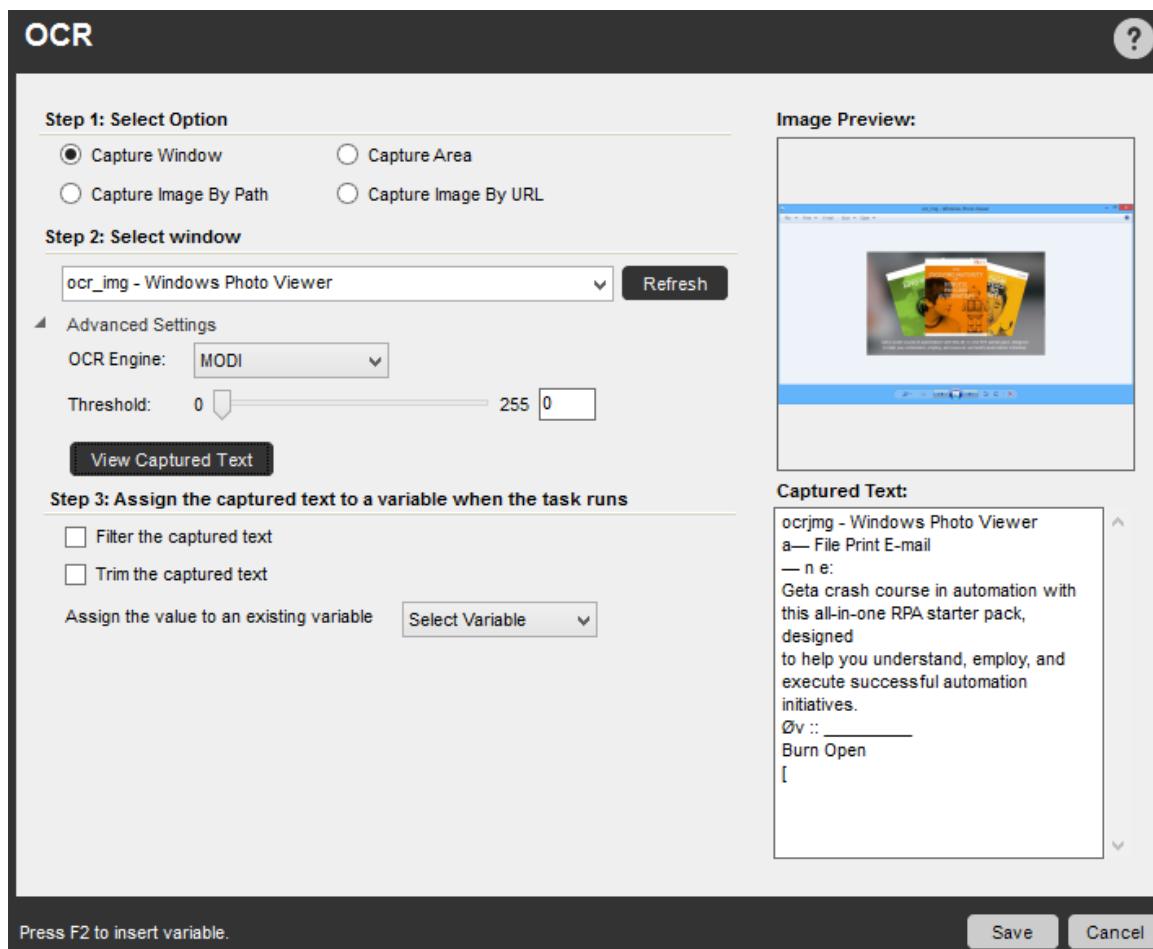
Using the OCR Feature

The OCR Feature command provides the following options:

- **Capture Window:** Specify the window title. Automation Anywhere captures the window as an image.
- **Capture Area:** Specify a specific area of the window to capture.
- **Capture Image By Path:** To extract text that is contained within an image that is stored on your local or network drive, specify the location of the file. The drive must be accessible when you run the task.
- **Capture Image by URL:** Specify a website URL that contains the image you want to capture.

Using Keywords to Identify Captured Text

To make specifying the target text to capture easier, use the Before and After keywords. For example, in the text string "Name: ABC Inc. Location:", to copy only "ABC Inc.," specify Before = "Location" and After = "Name:". You can also trim the captured text to remove leading and trailing spaces.

OCR

Step 1: Select Option

- ⦿ Capture Window   ◯ Capture Area
- ◯ Capture Image By Path   ◯ Capture Image By URL

Step 2: Select window

ocr_img - Windows Photo Viewer ▾   [Refresh]

◢ Advanced Settings

OCR Engine: MODI ▾

Threshold: 0 [slider] 255 [0]

[View Captured Text]

Step 3: Assign the captured text to a variable when the task runs

☐ Filter the captured text
☐ Trim the captured text

Assign the value to an existing variable   Select Variable ▾

Image Preview:

Captured Text:

```
ocrjmg - Windows Photo Viewer
a— File Print E-mail
— n e:
Geta crash course in automation with
this all-in-one RPA starter pack,
designed
to help you understand, employ, and
execute successful automation
initiatives.
Øv :: _____
Burn Open
[
```

Press F2 to insert variable.   [Save] [Cancel]

Also See...

- Using Image Recognition
- Running Tasks in Parallel Using the MultiTask Designer (Add-On Feature)
- Using SnapPoint
- Using the Report Designer
- Using the Workflow Designer
- Using the ROI Calculator

Using Image Recognition

Use the Image Recognition feature to compare many large images and automate application processes using visual elements.
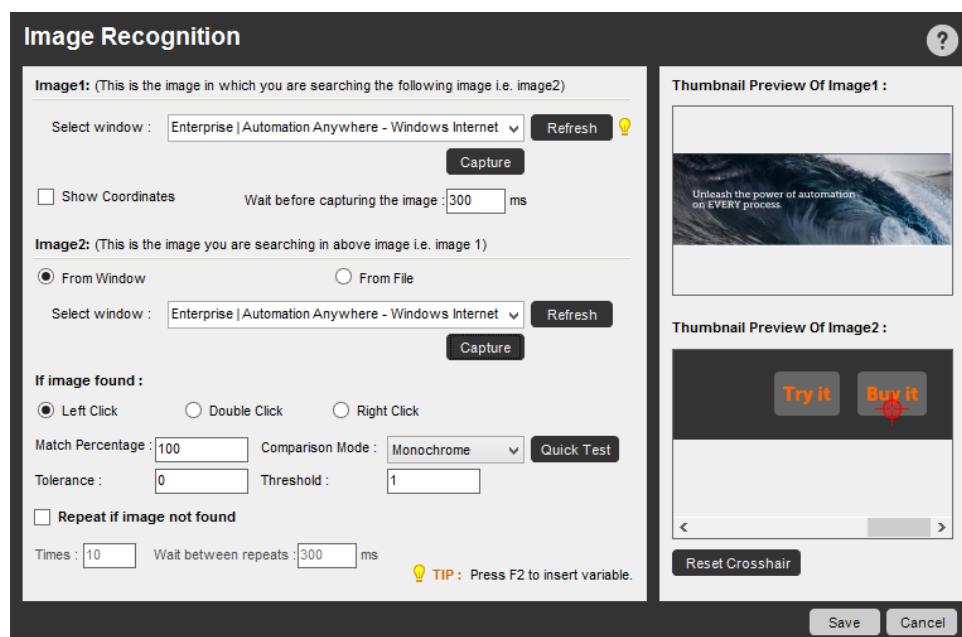
For automated testing, Image Recognition can be used for user interface validation. You can compare a current application screen to an expected application screen, as well as ensure that all user interface elements are valid.

Using Image Recognition, you can do the following:

- Locate an image within another image that is saved in a file or captured from a window at run time. The image is captured dynamically at run time in a window that you specify.

- Specify the minimum percentage match for which you want the command to return true.

- Left-click, right-click, or double-click the image if it is found within the larger image.

- Capture images from BMP, JPG, and GIF file formats.

- Compare images with two modes: Tolerance and Comparison Mode.

    - **Tolerance Mode:** Compares images having light and dark colors. It performs the color grading check.

        Specifying a 0% tolerance compares exact pixel color with the source image. Specifying a tolerance of 10% compares the 10% (plus/minus) color shade with the source image.

    - **Comparison Mode:** Provides the following options:
        - **Normal mode:** Compares the RGB components of the pixels with the source.
        - **Gray-scaled mode:** Converts the pixel to gray scale and then performs a comparison.
        - **Monochrome mode with Threshold:** Converts the pixel to either black or white and performs the comparison. The threshold value is used when converting the RGB or Gray-Scaled pixel to either black or white. A pixel value that is less than the threshold value is black; greater is white.

Also See...

- Using the OCR Feature
- Running Tasks in Parallel Using the MultiTask Designer (Add-On Feature)
- Using SnapPoint
- Using the Report Designer

- Using the Workflow Designer
- Using the ROI Calculator

Running Tasks in Parallel Using the Multi Task Designer

Automation Anywhere offers an add-on feature - the Multi-Task Designer that you can use to run multiple tasks in parallel.
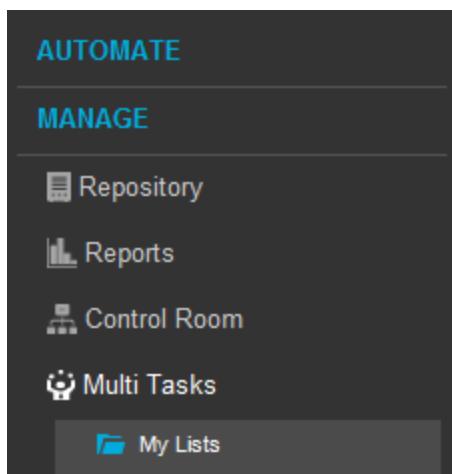
Using this facility, you can coordinate a set of automation tasks to run in parallel, on the go or setting the timing any way you like. Using the facility is simple, providing drag-and-drop operations made available in the Multi Task Designer window.

Purchasing a Multi Task Designer License

You can purchase licenses for the Multi Task Designer based on the number of concurrent tasks you want to run. If necessary, you can obtain a limited-time license key as well.

Enabling the Multi Task Designer

After you install the Multi Task Designer license key, the facility is enabled. A Multi Tasks tab is displayed on the left side of the main Automation Anywhere window under 'Manage'.
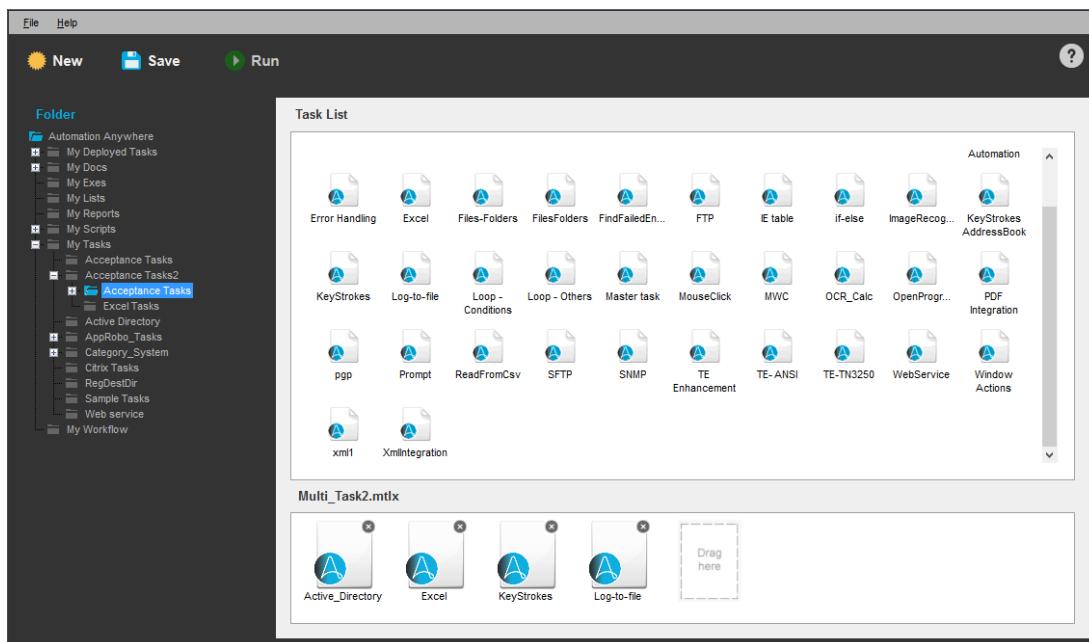
Using the Multi Task Designer

To use the Multi Task Designer, follow these steps:

1. In the main Automation Anywhere window, click on the Multi Task Designer tab.
2. In the Multi Task Designer window, all folders are displayed in the left panel.
3. To view all task files (.atmx files), click on a folder.
4. Create a new list of tasks that you want to run in parallel. Either drag a file or double-click on a file to add tasks to the list.
5. Save the Multi Task list by clicking the Save button. The saved list file has a file extension of .mtlx. Multi Task list files are stored in the My Lists folder.

6. Run the Multi Task list file. All tasks run in parallel, depending on the authorized concurrency count.
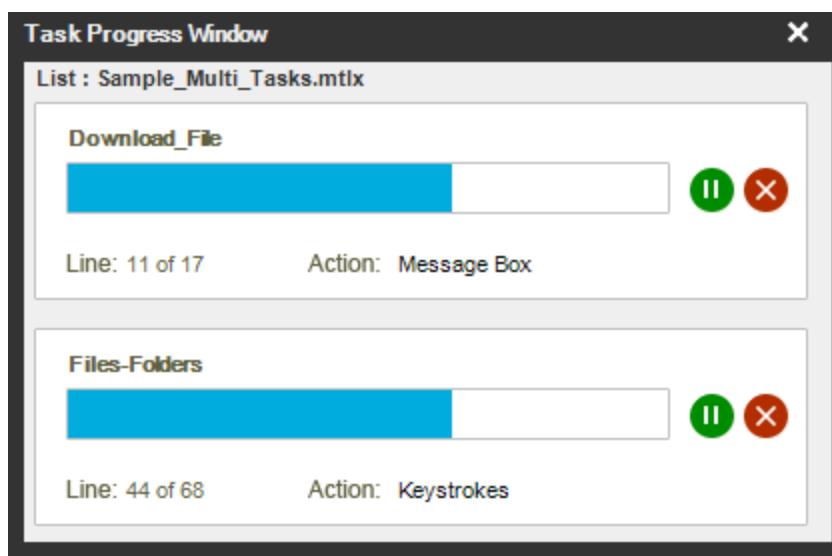
Removing a Task from the Multi Task List

To remove a task from the Multi Task list, select (highlight) a task from the list and click on the X in the file icon.

Running a Multi Task List File

To run your tasks in parallel, simply run the Multi Task list file that you've created by clicking on the Run button.

The Task Progress window is displayed, showing the first two tasks that are running in parallel. If you are running more than two tasks in parallel, a scroll bar is provided to view them.

Performing Actions on Multi Task Lists

You can perform the following actions on Multi Task list files (.mtlx) that are stored in the My Lists folder:

- **Run:** Runs a Multi Task list file from the My Lists folder.
- **Edit:** Enables you to edit a Multi Task list file. You can add or remove tasks from the list using the Multi Task Designer.
- **Delete:** Deletes a Multi Task list file.
- **Upload:** Uploads a Multi Task list file to the Server Repository, assuming proper privileges exist for the client.

Restrictions with Running Tasks in Parallel

The following restrictions exist for running tasks in parallel:

- Tasks can only be run in parallel using the Multi Task Designer.
- Task performance and completion depends on your available system resources. If system resources are fully consumed, tasks might stall or fail.

Recommendations when Running Tasks in Parallel

To ensure success, follow these recommendations when running tasks in parallel:

- Avoid running tasks in parallel that include the following interactive commands:
    - Insert Keystrokes
    - Insert Mouse Clicks
    - Message Boxes
    - Prompts
    - Image Recognition
    - Manage Windows Control

Avoid running two tasks in parallel that both access the same file or application (for applications that do not support multiple threads or sessions), such as a Microsoft Excel file. Opening, editing, and closing a single file using two parallel tasks can result in unpredictable results.

Supported Commands

The following commands are supported for running tasks in parallel:

- Excel commands
- Web Recorder commands
- Read from CSV/Text
- Database commands
- Log-to-file command
- String operations

Also See...
- Using SnapPoint
- Using the Report Designer
- Using the Workflow Designer
- Using the ROI Calculator

Using SnapPoint

Use the SnapPoint feature to add screen shots to an automation task or to recapture screen shots when a task is running.
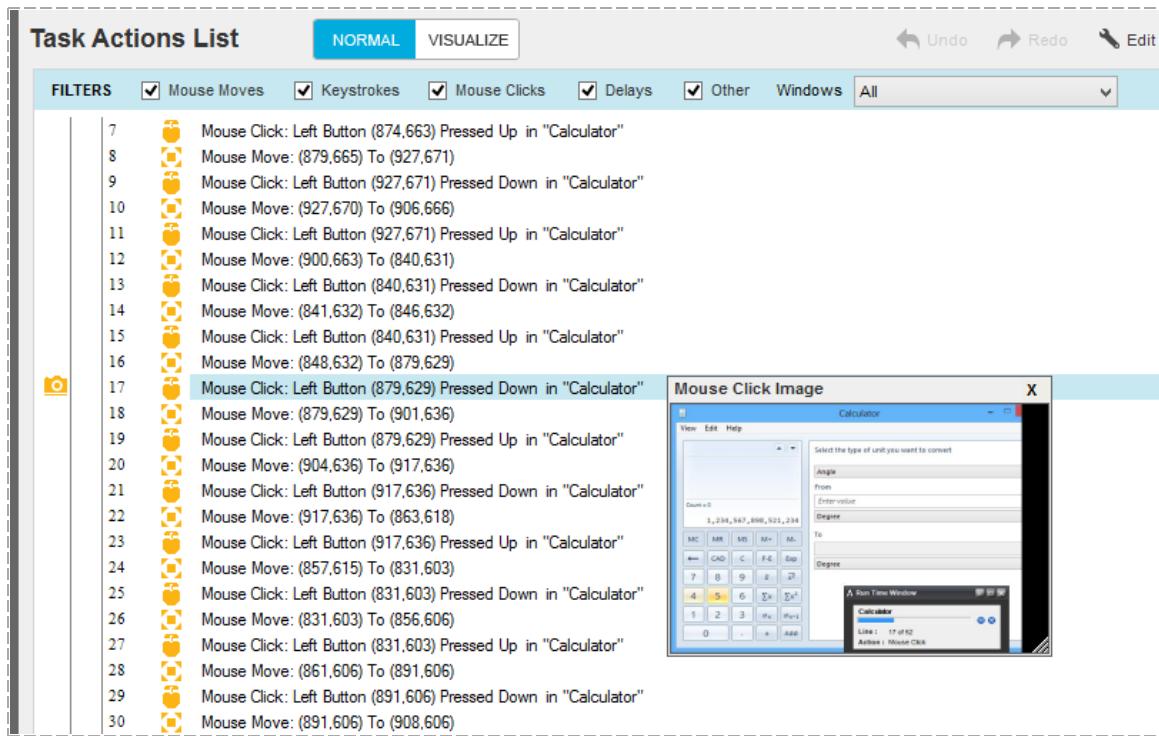
Capturing Images While Recording Tasks

Automation Anywhere provides a state-of-the-art technology for capturing and displaying screen shot images as you create automation tasks.

To capture screen shots, click on the Tools menu, select Options, and select Advanced Settings. Click on Capture Screenshots While Recording.

You can resize and move images inside the Task Editor to place them in a desired location. The captured image is the active window during recording, and shows you exactly where your mouse cursor is located.

For example, in the following screen shot, line number 24 in the task, Mouse Click: Left Button on 2 in Calculator, and the corresponding image showing the Windows calculator.

Using the Visualize Tab

Use the Visualize tab in the Task Editor to view a complete graphical layout of the current task. The following screen shot shows a task after having clicked on the Visualize tab. The example shows that you clicked Start, then Run, then OK in the Run window, then number '1' in the calculator.

Setting SnapPoints to Refresh the Images

When you create a task manually using the Task Editor, or you modify a recorded task, the images that you captured during recording might not remain the most current. Automation Anywhere provides a convenient way to refresh the images, called *SnapPoints*.

To set SnapPoints, click the Set SnapPoint button. You can now set SnapPoints in the column to the left of the line numbers, as shown below. When you run a task after setting SnapPoints, Automation Anywhere recaptures the images for those commands on which you set SnapPoints.

The images are captured after the task runs.

Commands that Support SnapPoints

The following commands support SnapPoints:

- Insert Keystrokes
- Insert Mouse Click
- Prompt
- Excel
- Web Recorder
- Window Actions
- Message Box
- Manage Windows Controls
- Screen Capture

Also See...

- Running Tasks in Parallel Using the MultiTask Designer (Add-On Feature)
- Using the Report Designer
- Using the Workflow Designer
- Using the ROI Calculator

Using the Report Designer

Use the Automation Anywhere Report Designer to run reports for your automation projects. These reports gives a graphical representation of automated tasks over a period of time. It also displays the ROI for an organization using Automation Anywhere products.

In This Section

- What is a Report?
- Running Reports
- Saving Reports
- Sending Email Notifications for Reports
- Scheduling a Report to Run
- Triggering a Report to Run
- Viewing a Report Graphically

What is a Report?

Reports are important feature to measure the success and calculate the ROI that Automation Anywhere brings to your organization with respect to time.
Automation Anywhere provides graphical reports that display the status of tasks and workflows, as well as the ROI over time.

You can view the following types of reports:

Task Run: Displays how many times a task has run successfully, and how many times it has failed during the period. You can view this report for:

- All Tasks saved in the default location, i.e., My Documents\Automation Anywhere\Automation Anywhere\My Tasks
- Tasks in a specified folder
- Selected tasks.

To graphically view your Task Run, see the Visualize Report.

Workflow Run: Displays how many times a workflow has run successfully and how many times it has failed during the period. You can view this report for:

- All workflows saved in the default location, i.e., My Documents\Automation Anywhere Files\Automation Anywhere Files\My Workflow
- Workflows in a specified folder
- Selected workflows

Task Timeline: Displays task history in a graphical time line. Select a task and time line to view the number of times a task was run or modified during the period. It also shows whether the task was created or deleted during that time. You can view this report for:

- All tasks saved in default location, i.e. My Documents\Automation Anywhere Files\Automation Anywhere Files\My Tasks
- Tasks in a specified folder
- Selected tasks

Workflow Timeline: Displays workflow history in a graphical time line. You can select a workflow and a time line to see the number of times the workflow was run or modified during the time period. It also tells you whether the workflow was created or deleted during that time. You can view this report for:

- All Workflows saved in default location, i.e. My Documents\Automation Anywhere Files\Automation Anywhere Files\My Workflow
- Workflows in a specified folder
- Selected workflows

ROI: Displays how much USD/Dollar Value has been saved by using Automation Anywhereduring the time period.

Visual Logs: Graphically displays tasks that have run by providing step-by-step SnapPoints for each action in a task.
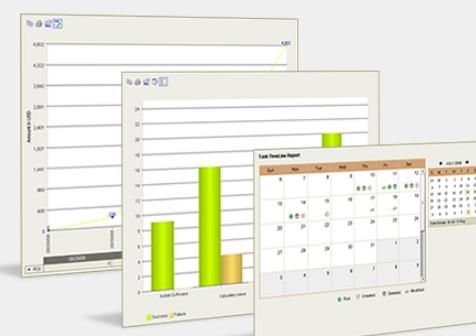
Also See...

- Running Reports
- Saving Reports
- Sending Email Notifications for Reports
- Scheduling a Report to Run
- Triggering a Report to Run
- Viewing a Report Graphically

Running Reports

How do I run a report?

Running a report is simple.

To run a report, simply specify your criteria in the Report Designer and click Run Report.

To run a saved report, double-click the report file in the Task List, or select the report in the Task List and click Run.

Also See...

- What is a Report?
- Saving Reports
- Sending Email Notifications for Reports
- Scheduling a Report to Run
- Triggering a Report to Run
- Viewing a Report Graphically

Saving Reports
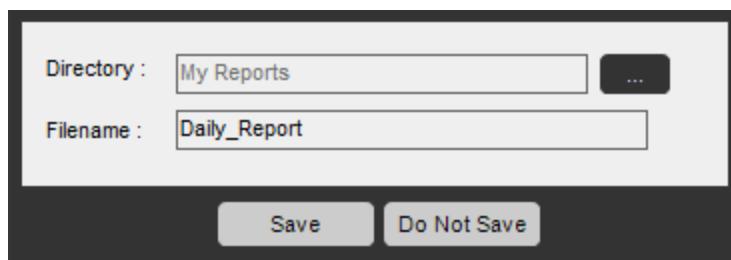
How do I save a report?

To save a report after specifying the criteria, click Save and Run Report.

Save and Run Report

Automation Anywhere stores your selections and saves the report as a template for future use.

If you want to save a report as an image, click on the File menu and select Save As Image.

Directory : My Reports ...

Filename : Daily_Report

Save   Do Not Save

Also See...

- What is a Report?
- Running Reports
- Sending Email Notifications for Reports
- Scheduling a Report to Run
- Triggering a Report to Run
- Viewing a Report Graphically

Sending Email Notifications for Reports

Using the Notification property, you can automatically send an email notification after a report completes.

The email includes the report as an image, as well as other helpful information.

Also See...
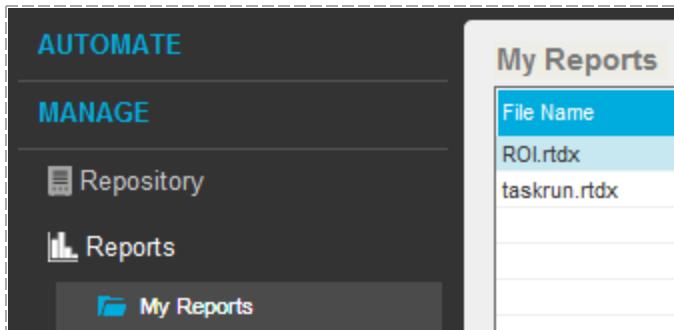
- What is a Report?
- Running Reports
- Saving Reports
- Scheduling a Report to Run
- Triggering a Report to Run
- Viewing a Report Graphically

Scheduling a Report to Run
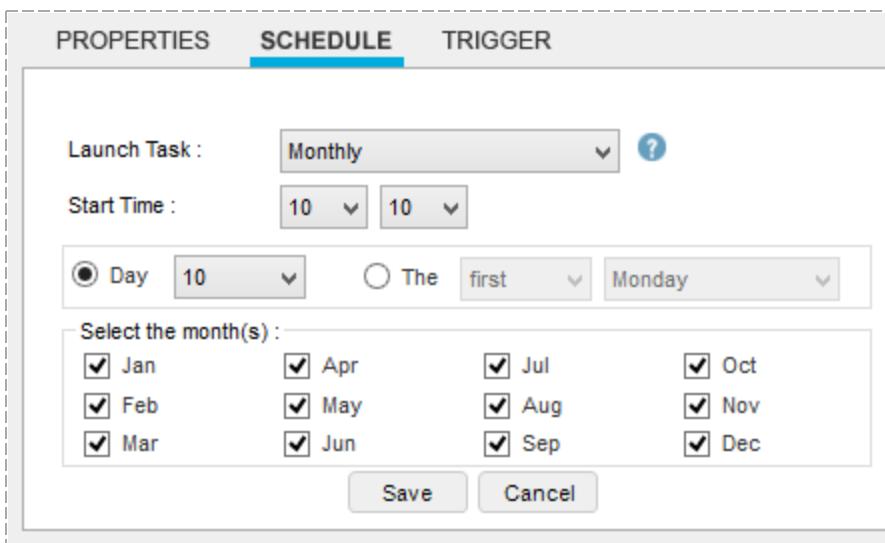
Using the Automation Anywhere scheduler, you can schedule a report to run at a specific time.

To schedule a report to run, follow these steps:

1. In the Task List, select the report.

2. Select Schedule.
3. Specify the scheduling parameters.

Also See...
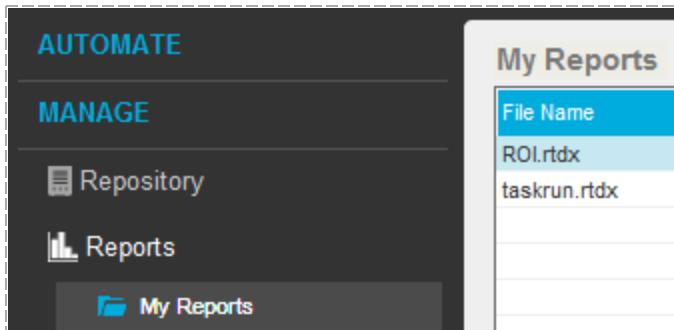
- What is a Report?
- Running Reports
- Saving Reports
- Sending Email Notifications for Reports
- Triggering a Report to Run
- Viewing a Report Graphically

Triggering a Report to Run
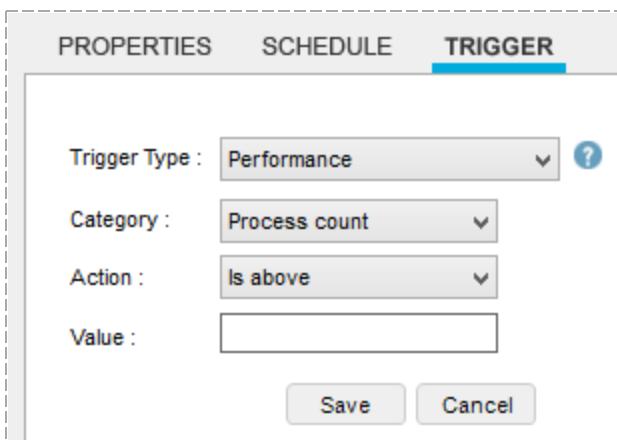
Using the Automation Anywhere trigger feature, you can run a report based on a trigger event.

To trigger a report to run, follow these steps:

1. In the Task List, select the report.

2. Select Trigger.
3. Define your trigger.

You can use the Notification property to send an email notification after the trigger event runs the report. For more information, see Sending Email Notifications for Reports.

Also See...

- What is a Report?
- Running Reports
- Saving Reports
- Sending Email Notifications for Reports
- Scheduling a Report to Run
- Viewing a Report Graphically

Viewing a Report Graphically
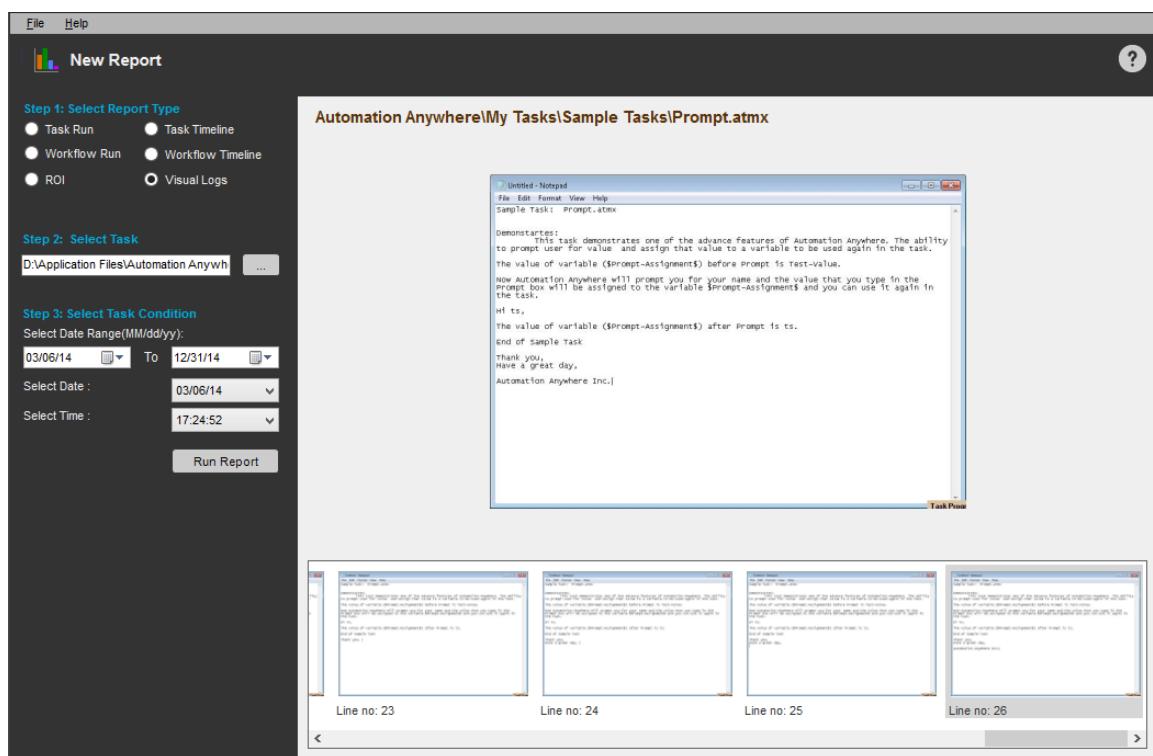
Automation Anywhere provides a step-by-step graphical view of all tasks run. To turn on the Visualize view, follow these steps:

1. Click on the Tools menu, and select Options.
2. Click on Advanced Settings.
3. Check the 'Capture Screenshots While Recording a Task' check box.
4. Click the Apply and OK buttons.
5. To view your task graphically, simply click on the bar representing the task using the Task Run report view.

If SnapPoints are not supported by the task, a message is displayed to notify you immediately.

Using the Visualize Report, you can:

- View the number of days that the task has run during a specific date range.
- View a specific day and the number of times that a task has run during that day.
- Compare your tasks using all of the saved SnapPoints in the task folders (..My Documents\Automation Anywhere\SnapPoints).

Also See...

- What is a Report?
- Running Reports
- Saving Reports
- Sending Email Notifications for Reports
- Scheduling a Report to Run
- Triggering a Report to Run

Using the Workflow Designer

To assist you in designing and improving your business workflows, Automation Anywhere provides a Workflow Designer tool. The Workflow Designer helps you build Workflow flow diagrams in a graphical environment. You can subsequently use these Workflow to build and run automated Workflows using Automation Anywhere.

Overview

A *Workflow* shows a high-level, graphical view of your business or IT Workflow, with a representation of conditional logic and Workflow flow. Some Workflows might be simple, some more complex.

The Workflow Designer provides an intuitive, drag-and-drop interface to help you quickly design your workflows.
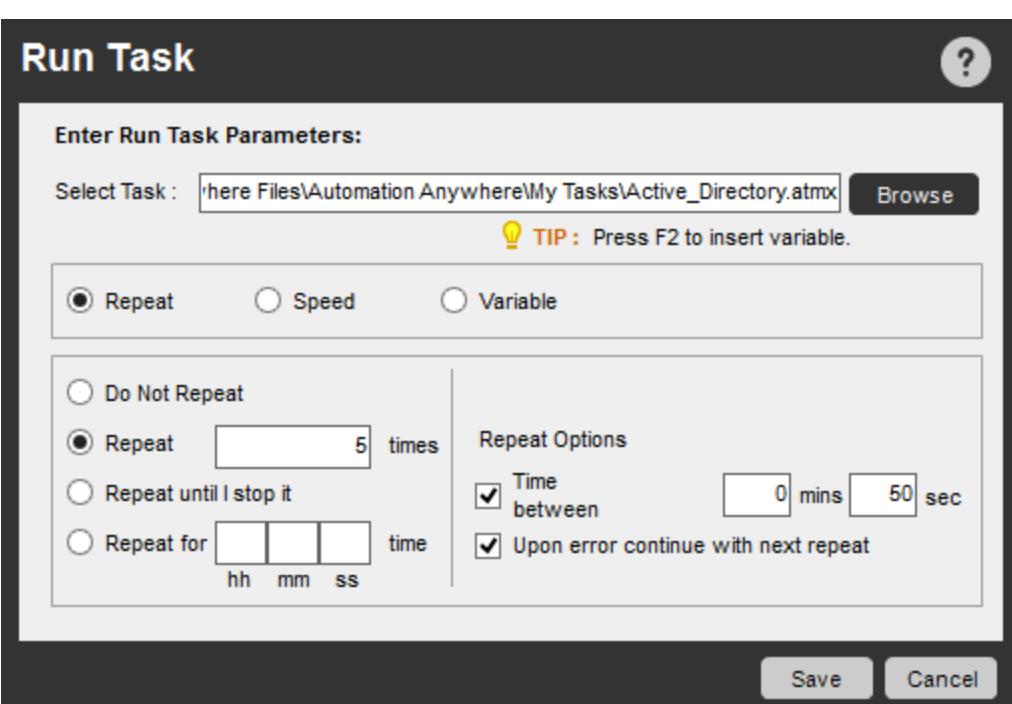
Creating a Workflow?

To create a new Workflow, follow these steps:
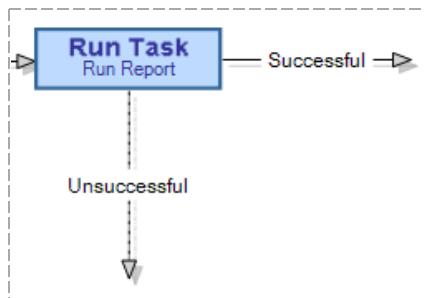
1. Click on the Tools menu and select Workflow Designer. The Workflow Designer provides the following design objects:

- **Start:** The starting point for the Workflow. Each Workflow can have only one Start object.

- **Run Task:** Runs a previously built task (.atmx file). Before proceeding to the next Workflow step, the Workflow stops until the Run Task command is completely finished.
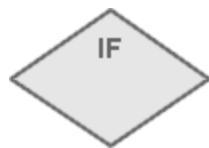
When you insert a Run Task object, it displays one Successful arrow and one Unsuccessful arrow. Both arrows are required for a complete Workflow. If you accidentally delete one of these arrows, move the mouse over the Run Task object and pull another arrow over to connect it with the desired object.

> Note: You can edit a task from the Run Task list by right-clicking the task and viewing the menu.

- **Conditional (If):** Provides an event condition on which the subsequent steps in the Workflow depend. Based on the result of the condition (True or False), the Workflow moves along the appropriate path to next steps.

  When you insert a Conditional object, it displays one True arrow and one False arrow. Both arrows are required. If you accidentally delete one of the arrows, move the mouse over the object and pull another arrow over to connect it with the desired object.

  Use multiple AND or OR conditions*- You can include 'AND' or 'OR' conditions when you wish to include multiple conditions in the same command.

  * Available from version 7.5.2 onward
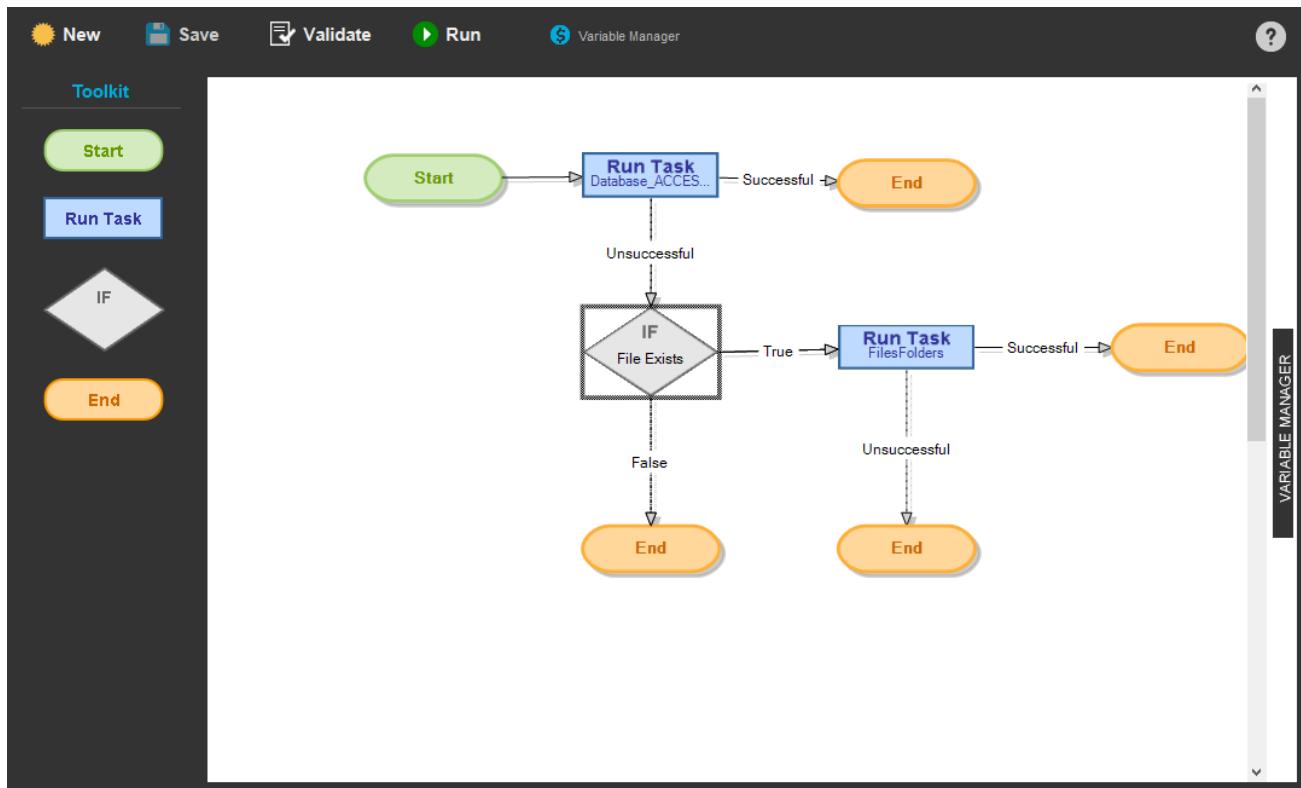
  Refer If Else Command to know how.

- **End:** The end of the Workflow. When the Workflow reaches this object, execution is stopped.

- **Arrows:** Connects objects and pathways in the Workflow. To insert an arrow from any object, move the mouse over the object until the mouse icon changes to a hand icon:
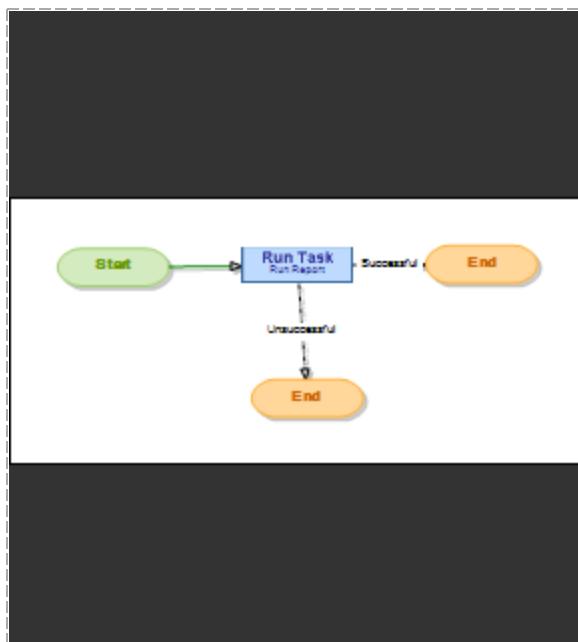
> 💡 Tip: Prior to running a Workflow or exiting the Workflow Designer, always validate the Workflow. Validation ensures that your Workflow objects are correctly connected and that errors do not occur.

Running a Workflow

To run a Workflow from the Workflow Designer, click on Run or double-click the Workflow file in the Task List.

When you run the Workflow within the Workflow Designer, Automation Anywhere indicates the path followed in the Workflow. For example, you can easily see whether a True or False condition occurs.

After a Workflow has completed a run, the Workflow Designer highlights the complete path in color.

Repeating a Workflow

To repeat a Workflow, use the Repeat property. The Workflow will run again.

Sending Email Notification for a Workflow

To send an email notification when a Workflow run is complete, use the Notification property.

Saving a Workflow as an Image

To save a Workflow as an image, click on the File menu and select Save as Image.

Printing a Workflow

To print a Workflow, click on the File menu and select Print.

Scheduling a Workflow to Run

You can schedule a Workflow to run using the Automation Anywhere Scheduler. Refer Scheduling Tasks to Run.

Triggering a Workflow to Run

You can use triggers to schedule a Workflow to run. Refer Scheduling Tasks Using Triggers.

Associating Workflow Variables to Task Variables

Workflow variables can be associated with various tasks. The associated variable is clearly shown in Run Task command.

Only type of variable association should be same, the names of variables can differ.

Steps to associate Workflow variable to task variables:

1. Click the Run Task command.
2. Select Task using Browse.
3. Select Variable option as action.
4. Create variable association section appears.
5. Select Worflow Variables from drop down list and associate them with the drop down of list for the Task Variables.

    Use the 'Quick Map' button if you have a large set of variables to associate *.
    Available from version 8.1 onward.
    Refer Run Task Command for details.

6. Associate workflow variables to task variables using Run Task > Variable option.
- Both drop downs will only display Value type of variables defined.

7. Use Add button to add more variable association of Workflow to Task.

8. Select an association and use Delete button, to remove it.

📝 Note:

1. If there are changes to the task variables type or source, the association will be invalid.

2. If there is change in application path of a task, the association will be updated auto-matically. However, one needs to manually Save the workflow.

3. The SAVE button is enabled for these workflows with an appropriate message.

Also See...

- Running Tasks in Parallel Using the MultiTask Designer (Add-On Feature)
- Using SnapPoint
- Using the Report Designer
- Using the ROI Calculator
- Using Variables

Using the ROI Calculator

Use the ROI Calculator as a planning tool that calculates your return on investment (ROI) on process improvement.
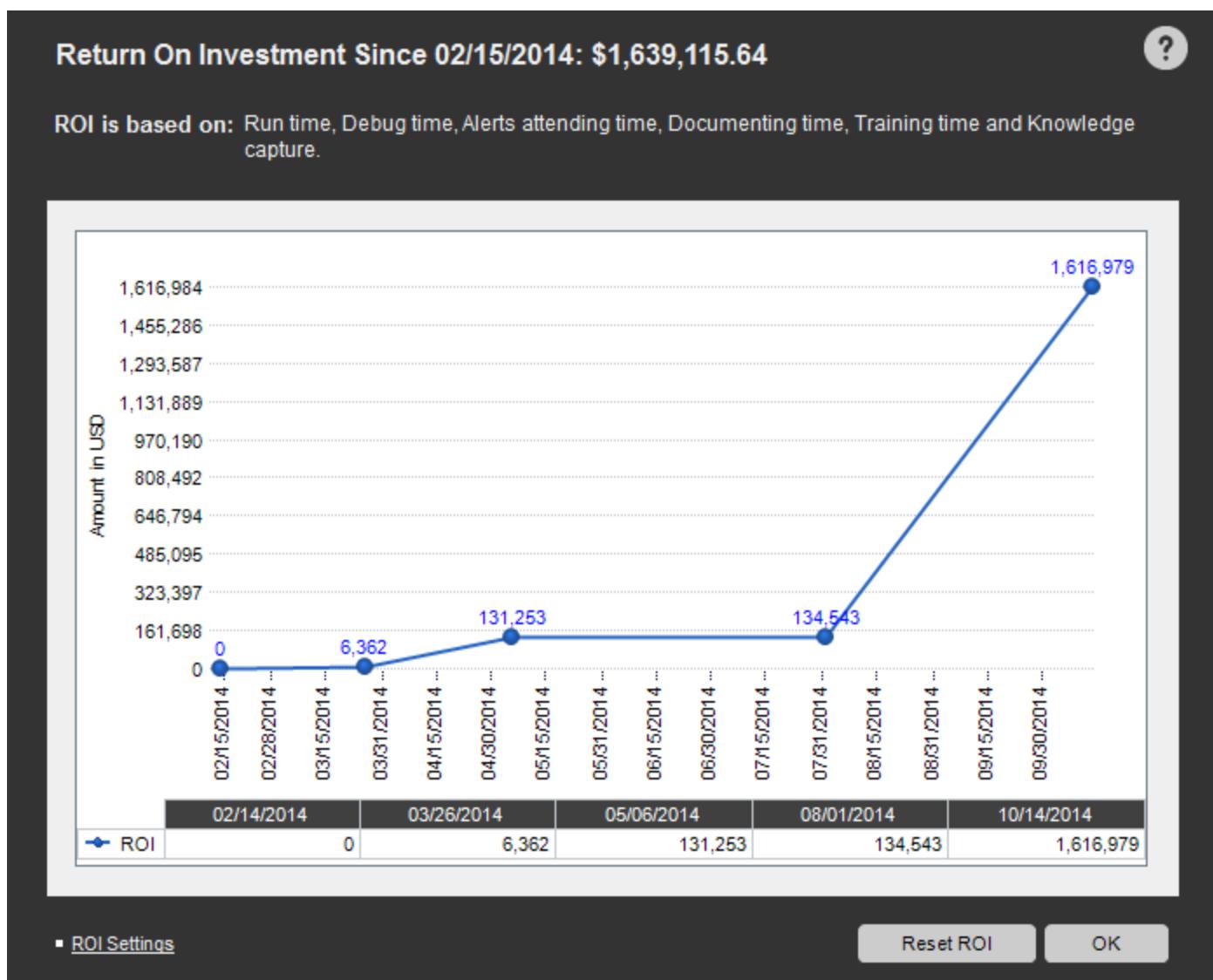
The ROI calculator makes the following calculations:

- Resource costs for time saved
- Reduced error rates
- Automated troubleshooting
- Automated documentation
- Resource realignment
- Training benefits
- Knowledge capture
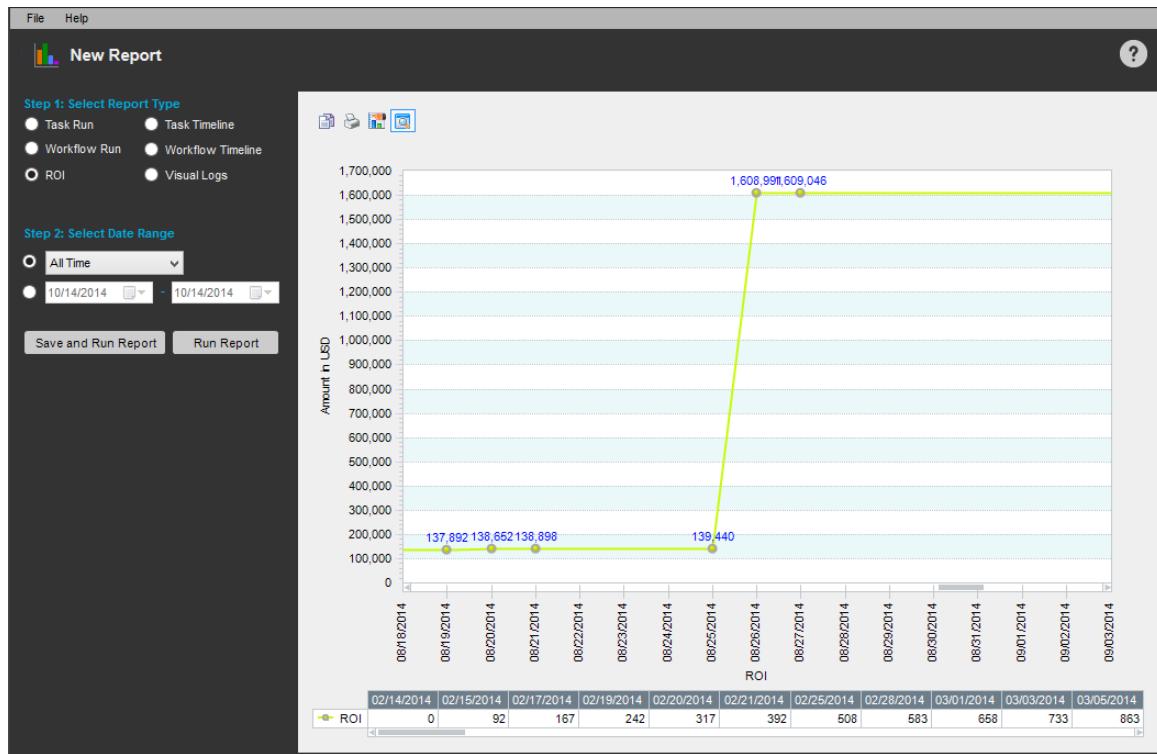
Automation Anywhere calculates the ROI based on the results of automating processes. The ROI is a cumulative dollar amount from running tasks.

As the basis for the ROI calculations, you can set the hourly personnel rate to calculate hourly savings. For more information, see the 'ROI Settings' in the ROI Calculator.

To make the ROI calculations easy to understand and useful in sharing, you can view the ROI as a graphical chart.

Return On Investment Since 02/15/2014: $1,639,115.64

ROI is based on: Run time, Debug time, Alerts attending time, Documenting time, Training time and Knowledge capture.

	02/14/2014	03/26/2014	05/06/2014	08/01/2014	10/14/2014
ROI	0	6,362	131,253	134,543	1,616,979

- ROI Settings

Reset ROI   OK

Note: You can reset the ROI at any time by clicking 'Reset ROI' button.

You can also generate ROI related reports using the Report Designer as shown:

Also See...

- Running Tasks in Parallel Using the MultiTask Designer (Add-On Feature)
- Using SnapPoint
- Using the Report Designer
- Using the Workflow Designer

Communicating with the Server

These topics provide information to help you communicate with the Automation Anywhere Small Business Server (or Enterprise Control Room) from your Automation Anywhere Client.

In This Section

- Registering a Client with the Server
- Uploading and Downloading Tasks to the Server
- Logging into the Server
- Comparing Files that Reside on the Client and Server

Registering a Client with the Server/Control Room

Applicable to Small Business and Enterprise editions for version 7.1 or less

Client registration is required to enable the Client to communicate with the Server. This topic describes how to register a new or existing Client with the Server.

If you have not previously registered the Client, register based on whichever option is applicable:

If the Automation Anywhere Client is Not Running

When the AA Client is not running, do the following:

1. Launch the Automation Anywhere Client.
2. On the initial Client Registration window, enter Server IP Address. If the Client is on the same system as the Server, enter 'localhost'.

3. Enter the server Port number that is being used by the server.
4. Enter the user name and password.
5. Click on Register.

If registration is successful, a message is displayed indicating "Your client is registered successfully."

If the Automation Anywhere Client is Already Running

If the Client is already running, you can register using two methods:

1. Using Server Communication
2. Using the Options features from Tools menu

Register using the Server Communication option

Follow these steps:

1. From the main Client screen, click on the Tools menu and select "Server Communication". Or Click directly on the Server Communication tab on the left panel. The program invokes the Client Registration window.
2. On the Client Registration window, enter Server IP Address. If the Client is on the same system as the Server, enter 'localhost'.
3. Enter the server Port number that is being used by the server.
4. Enter the user name and password.
5. Click on Register.

Register using the Options menu

Follow these steps:

1. From the main Client screen, click on the Tools menu and select "Options".
2. In the left pane of the Options window, select "Server Settings".
3. Enter Server IP Address. If the Client is on the same system as the Server, enter 'localhost'.
4. Select the server Port that is being used by the server.
5. Click on Apply and then click OK.

Re-registering a Client

If you need to re-register a Client, follow these steps:

1. From the main client screen, Click on the Tools menu and select "Options".
2. In the left pane of the Options window, select "Server Settings".
3. Click the Re-register.
4. Enter Server IP Address. If the Client is on the same system as the Server, enter 'localhost'.
5. Enter the server Port number that is being used by the server.
6. Enter the user name and password.
7. Click on Register.

Registration Details on Control Room/Server Management Console

For Enterprise Control Room Edition only

> Note: After successful registration, the Client Control Center tab in the Control Room indicates the following:

- The legend gets updated as shown:

    Refer Using the Client Control Center for details.

- The Client information and machine information gets updated:
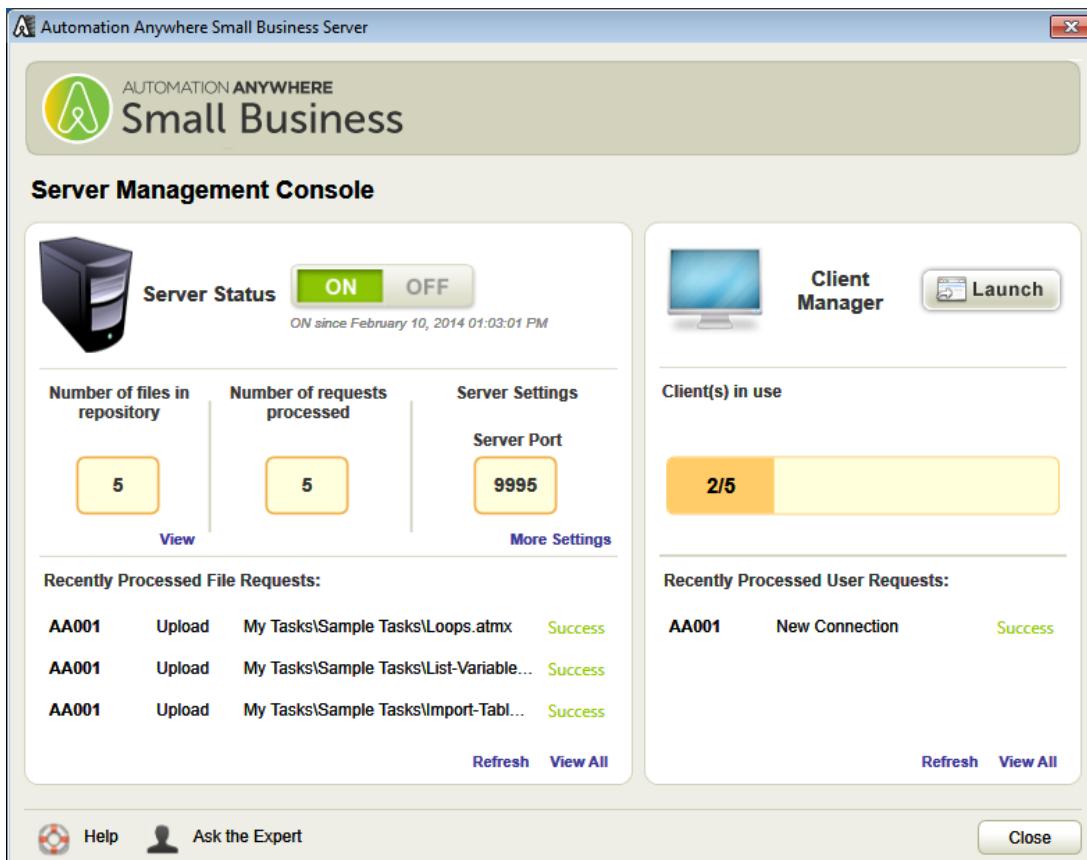
Refer Viewing Machine Information for details.
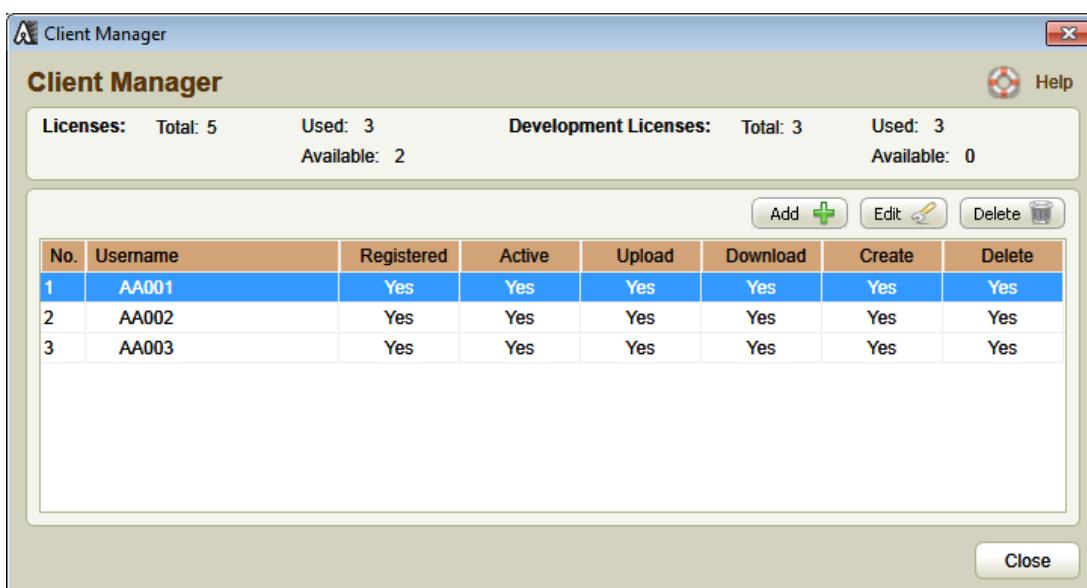
For Small Business Server Edition only

To register a Client with the Server, follow these steps:
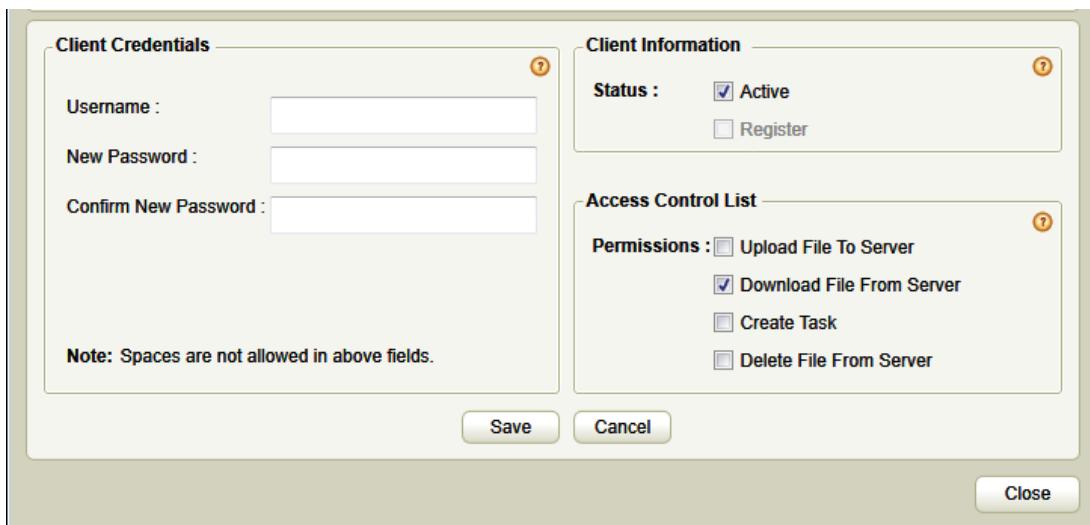
1. Launch Automation Anywhere Server by selecting Automation Anywhere Server from Windows System Tray. Right-click on it and select Automation Anywhere

- The Server Management Console is launched.

2. Click the Launch button to launch the Client Manager Window.
3. In Client Manager, click the Add button.

4. Specify the user name and password.

5. Enable any permissions for your Client user by using the options such as "Upload file to Server", "Download file to Server", "Create Task", and "Delete task from Server".

6. Save the settings.

7. Re-start the server by clicking on the 'Stop' button and then on the 'Start' button.

8. Close Automation Anywhere Small Business Server.

Also See…

- Logging into the Server
- Uploading and Downloading Tasks to the Server

Log in to the Server

Applicable to versions earlier than 7.5 for Small Business and Enterprise editions only.

For the client to communicate with the server, client authentication is required. This topic describes how to log into the server from a client.

This authentication is required only upon the first login.

By default, if an Active Directory user is already registered, the client logs in using these Active Directory credentials.

To change login information for a user, use **Re-login** to log in as a different client on the same machine.

> Note: For a client to successfully login, the Server must be up and running.

To log into the Server, follow these steps:

1. If you have not already registered previously, you need to register the user to enable communication with the Server.

    Refer Registering a Client with the Server/Control Room for details

2. In the Automation Anywhere Client Login window, click on Server Settings. Or click on Tools > Options > Server Settings.
3. Click on Re-Login.
4. Check the 'Use Active Directory credentials to login' check box to log in using the Active Directory user name and password.

- By default, the server domain is displayed for the current user. You can modify these credentials anytime.
- Either a message is issued indicating successful login or an error is displayed. When logged in, the user named is displayed at the bottom in the status bar.

Also See...

- Registering a Client with the Server
- Uploading and Downloading Tasks to the Server

Uploading and Downloading Tasks to the Server

Applicable to Small Business and Enterprise editions only.

This topic describes how to use the Server Communication feature to move automation tasks to and from the server.

The Automation Anywhere client provides an easy-to-use facility for communicating with the server.

To launch the Server Communication facility, click on the Server Communication tab on the main client screen.

Make sure that the client is already registered with the server.

Refer Registering a Client with the Server/Control Room for details.

You can use the Server Communication facility to perform the following tasks:
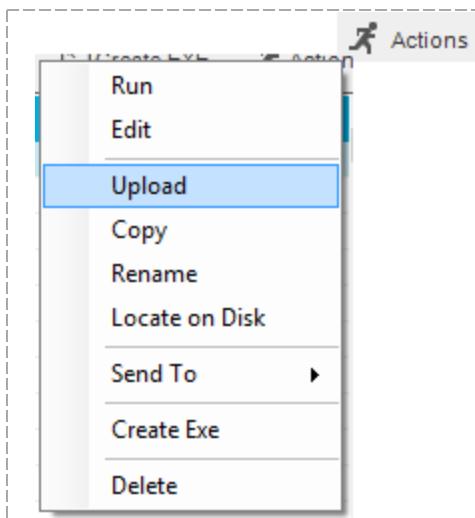
- Transferring tasks between the client and server.
- View the date and time that a task was last modified on the client or the server.
- View any new tasks that have been created on a client or uploaded to the server.
- Compare task files to identify differences between versions of the files that reside on the client and server.

Refer Comparing Files that Reside on the Client and Server for details.

Uploading a Task to the Server

You can upload tasks to the server (or the Control Room Repository Manager using any of the four methods:
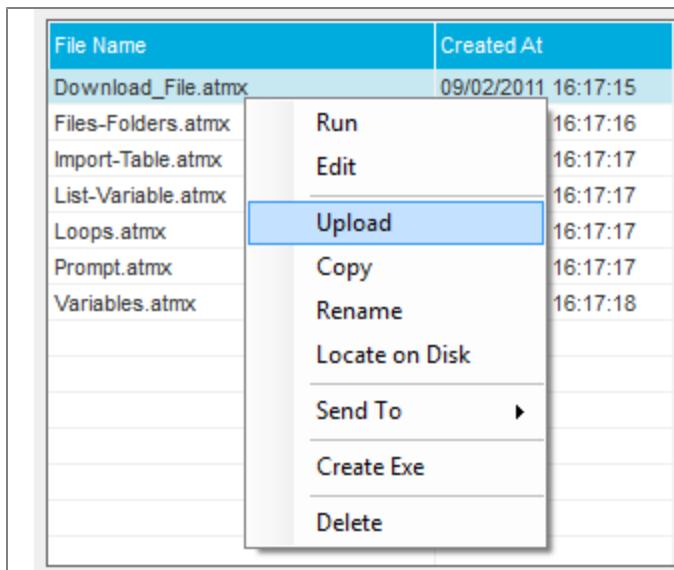
1. Selecting Upload from the Action button drop-down list
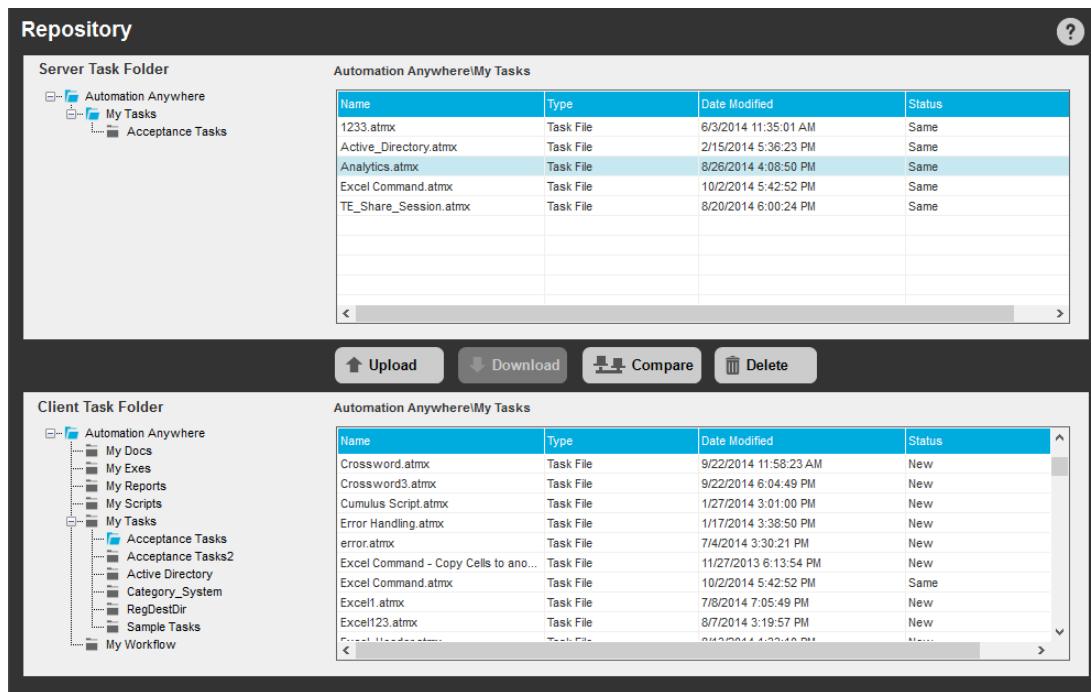
2. Clicking the 'Upload' button at the top right 'Task List' pane

3. Right clicking the selected task and choosing 'Upload'

4. Selecting the task from the Server Communication feature:

- When the task is successfully uploaded, a confirmation window is displayed. Click OK.
- The task is displayed in the Server Repository Manager.

Downloading a Task from the Server

To download a task from the Server (or Control Room Repository Manager), you must use the Server Communication facility. Follow these steps:

1. In the main Automation Anywhere window, click on the Server Communication tab. The Server Communication facility is displayed, with a split screen showing files that reside on the server as well as the client.
2. Highlight the task file that you want to download to the client.
3. Click the Download button. The task is copied to the client.

```
Note: Be aware that files that are downloaded to the client with same names will be overwritten.
Before downloading files, be sure that the latest version of the task is on the server.
```

Also See...

- Comparing Files that Reside on the Client and Server

- Logging into the Server

- Registering a Client with the Server

Comparing Files that Reside on the Client and Server

Your team of automation users can create, upload, edit, and run many tasks, moving task files between the client and server. Maintaining several copies of task files and having multiple users modify single tasks can require the need for users to view how and when a task file was last changed.

This topic describes how to compare versions of task files that reside on the client and server.

Using the Compare File feature, you can compare two different versions of a task file to identify any recent changes. You can view these differences by using the Task Actions List panes that are generated after running the comparison.

To run a comparison of two task files, follow these steps:

1. Launch the Repository facility by clicking on the Repository under Manage tab on the main client screen. Make sure that the client is already registered with the server.

2. Highlight the task file you would like to compare by clicking once on the file name, either in the Server or Client pane. In presence of a file with duplicate name in the Client or Server, the Compare button is enabled.

Compare

3. Click on the Compare button to run the comparison. A new window is launched, displaying the comparison results.

- If no differences between the files are found, the message "No differences found." is displayed at the top.

- If differences are found, the number of differences is displayed at the top.

- Differences in the task file actions are displayed in the following colors:

  - **BLUE:** Changed. Task actions that have been changed are marked in blue.

  - **GREEN:** Inserted. Task actions that have been newly inserted are marked in green on the client, and are shown as a blank line on the server.

  - **RED:** Deleted. Task actions have been deleted are marked in red on the server, and are shown as a blank line on the client.

Also See...

- Uploading and Downloading Tasks to the Server

- Logging into the Server

- Registering a Client with the Server/Control Room

Other User
Resources:
User forums
Ask the Expert
Open a support case
Automation Examples

Automation Anywhere Client FAQs

1. How does the Client communicate with Server?
2. Can I download/upload all types of documents?
3. What is the process of registration?
4. What do I do in case of an 'Account Lockout'?

How does the Client communicate with Server?

In the main Client window click on 'Repository' tab to communicate with the server. You can directly upload, download or compare any task on the server with the one on the client. Refer Uploading and Downloading Tasks to the Server for details.

Can I download/upload all types of documents?

Yes. You can download/upload all types of documents.

What is the process of registration?

The first time you start the client, it will ask you to login and when you click 'Login' (not 'Later') after inputting your credentials, you are registered with the server (Control Room/Server), automatically. Refer Login to Automation Anywhere Client for details.

What do I do in case of an 'Account Lockout'?

After the minimum allowable attempts have been exhausted for logging in to the Client account, it is locked for use. It can be unlocked only by the Control Room administrator, if a threshold value has been set in the Control Room. If the default domain policy is enabled, you are required to contact your System Administrator.

Refer Using Client Control Center for details.

Create Exe Feature FAQs

1. How do I create an Exe out of a task?

2. Is Automation Anywhere required to run the Exe created using the 'Create Exe' capability?

3. I have set most of the properties like email notification, speed, etc. for a task. If I create an Exe and delete the task, will the Exe retain all these properties?

How do we create an Exe out of a task?

This feature is available only in Automation Anywhere Premier and higher versions. For more information about Automation Anywhere versions check Compare Automation Software.

To create exe right click on any task and click on 'Create Exe' menu.

Is Automation Anywhere required to run the Exe created using the 'task to exe' capability?

No, even if you uninstall Automation Anywhere program the Exe will continue to work. The Exe is independent of Automation Anywhere.

I have set most of the properties like email notification, speed, etc. for a task. If I create an Exe and delete the task, will the Exe retain all these properties?

The Exe will retain all the properties set at that time. Once the Exe is created, it is independent of Automation Anywhere and its task. So it will retain all its properties.

Files Folders FAQs

1. Can I create a file with any extension?

2. If I use a Files / Folders command like "Delete Files 'c:\MyPrgs*.*' " , will this delete all the files in the sub-folders too?

3. Can I get list of all the file names in a folder to perform some operation on it?

4. Can I provide value for file/folder name when the task runs?

Can I create a file with any extension?

Yes, you can create file with any extension using the 'Create File' sub command in the Files/Folders command.

If I use a Files / Folders command like "Delete Files 'c:\MyPrgs*.*' ", will this delete all the files in the sub-folders too?

No, this command will delete all the files in the current folder only. In the above example it will delete all the files in 'c:\MyPrgs' directory only.

Can I get all the file names in a folder to perform some operation on it?

Yes, you can use the "Loop for Files in a <folder>"command.
Here $FileName$ will have next filename inside the specified folder in each Loop execution. You can use this System variable to perform any operation inside the Loop - End Loop block.

Can I provide value for file/folder name when the task runs?

Yes. Automation Anywhere has a 'Prompt' command to provide value at run time. You can provide value for any window, prompt for file/folder and even prompt for yes/no as well.

IF Condition FAQs

1. Can I execute certain commands only if a window exists?

2. Is it possible for me to know if one of my software is running or not? I want to do certain action only if that software is running.

Can I execute certain commands only if a window exists?

Yes. There are number of conditions in IF command. One of them is 'If Window Exists' .

Is it possible for me to know if one of my software is running or not? I want to do certain action only if that software is running.

Yes, you can use If Condition (If Application Running) to see if any application is running or not. You can specify the exe name that you want to test in the If condition. For e.g. consider the following script
If Application Running ('F:\WINNT\system32\calc.exe')
   Comments: Add your commands when the Calculator program is running
End If

Internet FAQs

What is E-mail Notification?

You can use E-mail notification when you want to be notified that a task has finished running. An e-mail will be sent to the addressed specified along with the status of the task that was running and few other properties.

What is this 'Email Settings' displayed in the Notification tab in the properties window?

This Button will open up the Options user interface that allows you to set the outgoing mail server, its port number and Exchange Server authentications, if any. Every mail that you send or receive requires the mail server. You have to obtain this information from your Internet E-mail provider or you may obtain it from the configuration section of your e-mail program.

Does your automation software provide any facility to send an email?

Yes, Automation Anywhere provides a command 'Send mail'. You can use this command to send e-mails to anyone. Note that this command will get information about the e-mail Server, its port number and Exchange Server authentications, if any, from the Mail Setting in the Options menu every time a task runs.

Loops FAQs

1. What is a Loop Command?

2. I want to do particular action on all the folders inside a folder. How do I do that using your automation software?

3. Can I get all the file names inside a folder using the Loop command?

What is a Loop Command?

A loop command allows repeated execution of a group of commands when these commands are placed inside the 'Loop " End Loop"' block. The number of the times these commands are executed will depend on the type of Loop command selected. The 'Loop - End Loop' block might also contain within it another 'Loop - End Loop' block i.e. nested loop commands.

I want to do particular action on all the folders inside a folder. How do I do that using your automation software?

You can use the Loop command (Start Loop 'Each folder in '<folder>') in Automation Anywhere. If there are 'n'™ folders in the specified folder then this loop will be executed 'n'™ times. You can get each folder's name using the $FolderName$ system variable inside this loop block.

Can I get all the file names inside a folder using the Loop command?

Yes, you can get the file name using the System variable $FileName$ inside a specified folder by using the Loop command " Start Loop'Each File' in '<folder>'.

Manage Windows Control FAQs

How to use and manage Windows Controls to get check-box or radio button status in Automation Anywhere?

Manage Windows Controls command allows you to work with windows controls like check-boxes and radio buttons. With this command you can find out if the radio button or check box is checked or not. Automation Anywhere supports only standard Windows check-boxes and radio buttons.

Click on Manage Windows Control in Task Editor and select the appropriate radio button. Select the window to capture the check box or radio button. The value captured can be assigned to a variable.

Does a task recorded using MWC work in the background?

Yes. A task recorded using MWC works in the background.

Why aren't some controls highlighted when I try to capture them using MWC?

We capture standard Windows control using MWC. Client controls and Java are not supported by MWC. In such cases, controls are not highlighted.

Moved Icons-Menus FAQs

1. I recorded a task that opens a program from Start Menu > Programs and while running the task, the menu to open that program has moved. It clicks on a wrong program name and hence opens a wrong program. Do I need to re-record the task?

2. Will my task run correctly even if files, program shortcut icons in the Quick Launch and the Desktop have moved?

I recorded a task that opens a program from Start Menu > Programs and while running the task, the menu to open that program has moved. It clicks on a wrong program name and hence opens a wrong program. Do I need to re-record the task?

No, the SMART Automation Technology built into Automation Anywhere provides a powerful feature that lets you use your original task. You can check an option from Tools > Options >'Move Icons/Menus' . After selecting this option when you run a task it opens the same program that was opened while recording even if the menus have moved.

Will my task run correctly even if files, program shortcut icons in the Quick Launch and the Desktop have moved?

Apart from a few exceptions, opening of most of the files and shortcut icons from both Quick Launch and Desktop will run correctly if the icons have moved from their original location. Note that, the 'Move Icons/Menus' should be checked in Tools > Options menu for this feature to be enabled.

Report Designer FAQs

1. What is Report Designer?
2. Can I print my report generated by Report Designer?

What is Report Designer?

Report Designer allows you to create your custom reports. Automation Anywhere features Task Run (i.e. whether they were successful or not), Task Timeline(i.e. run, created, modified and deleted), Workflow Run, Workflow Timeline, ROI and Visual Logs reports.

Reports can be generated by day, week, month as well as 'all time'. You can also generate a report for a custom date range.

Can I print my reports generated by Report Designer?

Yes. You can print a report generated by Report Designer directly from Automation Anywhere. Just click on the Print logo in the top left corner of the graph.

Recordings FAQs

How do I record a task using Automation Anywhere?

You can easily create new task using SMART task recorder. Click 'RECORD'; perform the activities on your computer that you want to record, click 'STOP' and then 'SAVE' the recorded activities to a task. You can playback your recorded actions by selecting the task and then click the 'RUN' button. For further details refer to the Help file.

Does it matter if the window size and locations of my applications change between recording and running?

No. Automation Anywhere's SMART Automation Technology automatically adjust to changes of window size and location.

Do I need to finish all my work in one application before moving onto the next application or can I switch back and forth between various applications while recording?

You can switch back and forth between various applications while recording.

Can Automation Anywhere work if computer is locked? Can I record a task to unlock the computer?

Yes. Automation Anywhere offers secure Auto Login technology. Auto Login allows scheduled tasks to run even when computer is locked. When you schedule a task you can enable Auto Login capability. If Auto Login in enabled, and if computer is locked, Automation Anywhere will unlock the computer, run the task and lock it back.

Is it possible to execute a task in stealth mode when computer is locked?

Yes. To enable the task to run in stealth mode check Properties->Security-> Run this task in stealth mode option. This will hide the execution of the task. To allow the task to run when computer is locked, enable Auto Login capability. Auto Login allows scheduled tasks to run even when computer is locked. When you schedule a task you can enable Auto Login capability.

An additional security option can also be set, where if you want to lock mouse and keyboard while task is running, just select 'Disable mouse and keyboard for this task' option from Properties->Security.

Can I stop the task while it is running?

Yes. Hold the Escape Key for 2-3 seconds.

Can I pause the task while it is running?

Yes. Press 'Pause' on your keyboard. This will temporarily stop the execution of the task. Click on the 'Resume' button that will appear at the bottom right hand corner of your screen to continue running the task from where it was paused.

Can my recorded activity run in background, while I have control of mouse and keyboard so that I can do other things?

One of the distinguishing features of Automation Anywhere is its ability to run tasks in background. Advanced technologies like Web recorder, Object recorder and hundreds of powerful actions allow you to run many of your tasks in background. Few tasks like task recorded with standard recorder requires control of mouse and keyboard and therefore cannot run in background. Few other obvious exceptions include commands that requires screen, like taking screenshots or comparing images on the screen, but a vast majority of actions allow the task to run in the background.

I use remote access software to access a remote machine. Can I record a task that performs activities using remote access software?

Yes. Automation Anywhere supports all the remote access software.

I recorded task on 'abc.xls' but I want to run it on xyz.xls, how do I do that?

You can run the same task with different files of the same program. By default the task will only run on the specific file(s) it was recorded on. However you can check 'Enable this task to run with other similar files and windows' option under the Properties>General tab. Now for example if you recorded a task on abc.xls, you can run this task on def.xls, xyz.xls or any other *.xls files

Is it possible to password protect my task?

Yes, it is possible to password protect a task. While saving the task click on the Password Protection checkbox and specify the password. Alternatively, you can password protect a saved task by going into the Security tab in the main screen of Automation Anywhere.

Scripting FAQs

1. I have been using VBScripts and JScript to accomplish some of my routine work. Can I somehow reuse them in your auto-mation software?

2. How do I know if my Script ran without any Error?

I have been using VBScripts and JScript to accomplish some of my routine work. Can I somehow reuse them in your automation software?

Yes. Not only you can integrate disparate scripts like VBScripts & JScripts within an Automation Anywhere task but you can use in-built automation services like repeat, notification, etc., on VBScript & JScripts as well. For more information about Automation Anywhere check Automation Software.

Automation Anywhere Editor has a command 'Run Script' . You can use this command to Run your VBScript or JScript files from Automation Anywhere. You can even pass arguments to your VBScript and JScript files as you might have been doing it to run it through command prompt. You can also have return values, if any, to your VBScripts and JScript files.

How do I know if my Script ran without any Error?

You can use the 'If Script Successful' sub-command inside the 'If command'. This command will run the script and will execute the 'If - End If' block only if the script ran successfully.

Tasks FAQs

Can I run many tasks in a batch?

Yes, You can create a new task in which you can use 'Run Task' command to run your individual tasks.

When I use "Run Task" command, how do I know if the task successfully completed?

You can use the 'If Task Successful' sub-command inside the 'If command'. This command will run the task and will execute the 'If - End If' block only if the task ran successfully.

Triggers FAQs

Can I execute a task based on availability or non-availability of any file, folder, performance or service?

Yes, it is possible to execute a task with Triggers, based on availability or non-availability of any file, folder, performance or ser-vice. There are triggers based on Window (open or close), File (create, delete, renamed or modified), Performance (CPU usage, free disk space, process count) and Process and Services.

Is there a way to perform a task when a new email arrives?

Yes. Automation Anywhere has a trigger functionality that can perform any task when a new email arrives. You need to set the'E-mail Message Trigger' on the task.

Variables FAQs

How do I use a variable in the task?

You can use variable inside different commands like'Insert Keystrok', 'Open Program/File','Files/Folders'. In all the commands variable names are specified by enclosing the variable names in between the $ sign. For e.g. $Password$

If I create a variable for one task, can I use it in another task?

Variables created for one task are only available locally for that task and not for any other task. But, if you have upgraded from a lower version to version 5.5 and above, all the variables created in the lower versions will be available to you as global variables that can be used with any task.

List variable can have many different text values. When I use the list variable which value will be executed?

Every time a task comes across a list variable during the playback, the next text value in the sequence of the list variable is used. If the list variable reaches the last text value then its subsequent reference will return the first text value.

Is it possible to store special characters like Tab key in a variable?

Yes, you can store any special character as a text value in a variable.

I have to run a task that needs to use a different value for a variable in every repeat. How do I do that with your automation software?

Automation Anywhere provides "Loop â€" End Loop" commands. You can place your repetitive task inside a Loop â€" End Loop block. You can use a List variable inside it for data that keeps on changing with each loop.

What are these System Variables $CurrentDirectory$, $FolderName$ and $FileName$?

These system variables are used with commands 'Loop for Folders in a folder' or 'Loop for Files in a folder'.

Consider a command in the task - "Loop for Folders in a <folder>".
 Here the <folder> is the specified folder on which the Loop will execute. Its full path is available at execution time by the system variable "$CurrentDirectory$"
 If <folder> has 'n' number of folders, the loop will be executed 'n' times. Each time the $FolderName$ variable will have the next folder's name inside the specified folder.

 Now if the task is like "Loop for Files in a <folder>".
 Then $FileName$ will represent the next file name inside the specified folder.
 Outside the loop $CurrentDirectory$, $FolderName$ and $FileName$ doesn't have any usage.

Visualize Technology FAQs

What is Visualize Technology in Automation Anywhere?

Automation Anywhere Premier has an in-built Visualize Technology that allows for pictorial and storyboard views of the automated task. Visualize Technology is used in the different views in Task Editor, to capture images on demand with SnapPoint and Image Recognition with the IF command.

What is SnapPoint in Automation Anywhere?

In Automation Anywhere Premier when a task is recorded the images of all mouse clicks and keystrokes are captured automatically. But when a task is created in Task Editor the images need to be captured on demand. SnapPoint allows you to capture images on demand; just enable SnapPoint and set it for the commands that you want.

What is Image Recognition in Automation Anywhere?

Image Recognition is another feature of the Visualize Technology available in Automation Anywhere Premier. Image Recognition allows the recognition and comparison of images. It allows your automated task to "see".

Web Recorder FAQs

What is a Web Recorder?

Web Recorder reliably and accurately records and automates web based tasks. Automation Anywhere Web Recorder understands all the web controls. You can open or close browser, perform clicks, fill forms, import data from the web, download files, extract data, extract source, check for broken links and much more.

What is web data extraction? How do you extract data with Web Recorder?

Extracting structured or unstructured data from the Internet automatically with the help of scripts is web data extraction or screen scraping. Automation Anywhere as a data extraction or screen scraping software is both flexible to suit any kind of web technology or page format and is also non-intrusive.

See two demos below to see how to use Automation Anywhere web recorder for data extraction.
Demo 1: Web data Extraction using intelligent pattern matching technology
Demo 2: Web data Extraction demo 2

Learn more on Web Data Extraction and Screen Scraping.

How do you check for broken links with Web Recorder?

To check for broken links with Web Recorder, go to Task Editor and select Web Recorder -> Find Broken Links command. Enter the URL and specify if you want the entire website to be checked or just the specified webpage. Enter the path of the file to save the results and click Save. Automation Anywhere will check for broken links as specified. You can schedule to run this task at pre-defined time.

What is Manage Web Control?

To capture image links, radio buttons, text boxes, check boxes and other web controls Automation Anywhere Web Recorder -> Manage Web Controls allows capture of web controls and assign them variables.

Can Automation Anywhere automate web based processes? If yes, what type of websites can be automated?

Yes. Automation Anywhere can automate web based processes with its Web Automation capability. You can automate any website, even complex websites that use javascript, AJAX, Flash or iFrames.

Learn more on Web Automation

Workflow Designer FAQs

What is Workflow Designer?

Workflow Designer allows you to create an easy to understand, intuitive graphical representation of a IT or a business processes. Automation Anywhere Premier allows any user (Business or IT) to easily create end to end business workflows.

How do I use Workflow Designer?

Open Workflow Designer (Shortcut Ctrl+K). Drag-n-drop the objects from the toolkit to design an entire business process. Workflow Designer provides four objects to design a Workflow:

- Start
- Run Task
- Condition (IF)
- End

How do I run workflows?

Running a workflow is very easy. Just click on the Run button in the Workflow Designer. The other option is to click on the workflow in 'MyWorkflow' file in the Task List and click Run.

On running the workflow through the Workflow Designer, it highlights the part of the workflow that is being carried out. Once, the workflow has completed running, it highlights the complete path taken.

Other FAQs

1. I recorded a task that opens a program from Start Menu > Programs and while running the task, the menu to open that program has moved. It clicks on a wrong program name and hence opens a wrong program. Do I need to re-record the task?
2. Will my task run correctly even if files, program shortcut icons in the Quick Launch and the Desktop have moved?
3. Can I get the snapshot and/or log data into file when error occurs?
4. Can I execute DOS commands on remote terminal?
5. My scheduled tasks do not run?
6. My task is big. How do I test only a few commands that I have added in the task?
7. I want to copy a command and use it at many places. Do I need to create the command every time?
8. I have created a keystroke which contains complex sequence of keys, which I use repeatedly in many of my task. Do I need to copy the keystroke every time I need to create a new task?
9. I have recorded a task that can be repeated as it is; only one text has to be replaced every time. I need to enter this text while running the task. How do I do that in your automation software?
10. Can I edit multiple commands at the same time? Is it possible to replace window title in all commands?
11. What happens to a triggered or scheduled task if a high priority task is already running?
12. Can I use Automation Anywhere as a macro recorder?
13. Can I use Automation Anywhere to integrate Excel commands with other applications?
14. How does Automation Anywhere carry out database automation?

I recorded a task that opens a program from Start Menu > Programs and while running the task, the menu to open that program has moved. It clicks on a wrong program name and hence opens a wrong program. Do I need to re-record the task?

No, the SMART Automation Technology built into Automation Anywhere provides a powerful feature that lets you use your original task. You can check an option from Tools > Options >'Move Icons/Menus' . After selecting this option when you run a task it opens the same program that was opened while recording even if the menus have moved.

Will my task run correctly even if files, program shortcut icons in the Quick Launch and the Desktop have moved?

Apart from a few exceptions, opening of most of the files and shortcut icons from both Quick Launch and Desktop will run correctly if the icons have moved from their original location. Note that, the 'Move Icons/Menus' should be checked in Tools > Options menu for this feature to be enabled.

Can I get the snapshot and/or log data into file when error occurs?

Yes, Automation Anywhere Premier has advanced error handling capabilities. You can select action like Continue or Stop Task. You can specify error handling options like Take Snapshot, Run Task, Log data into file, Send Email and Variable Assignment.

Can I execute DOS commands on remote terminal?

Yes, using Terminal Emulator command you can perform such activities. Terminal service must be on at the remote side and policy service must be set to classic. You can upload any file on FTP, Start/Stop services and execute DOS commands on remote terminal.

Learn more about Terminal Emulator.

My scheduled tasks do not run?

If your scheduled tasks don't run, follow the simple checklist to fix the problem.

1. Ensure that you have provided the username/password: For security reasons Windows XP requires your Windows XP user-name & password to run scheduled task. For more information on what to do if you don't have password read Why user-name/password? On any other Windows operating system, if your account requires a password to log on, then you need to provide that password while scheduling the task, otherwise you don't need to provide a password.

2. If you have multiple account ensure that you have provided the correct username/password: If you have multiple accounts on same computer like one domain account and one local account, please make sure the username that you have provided is the same username you have used to login to the computer. The tasks will not work if you scheduled the task using one user Account and now you are logged into another user Account. Automation Anywhere automatically fills the username that you are logged in as.

3. Check Status in Control Panel->Scheduled Tasks window: Please go to Control Panel->Scheduled Tasks window. Select the task that corresponds to your task (it start with your task name) and right-click and select 'Run'. See if it runs. If it does not run please see what the 'Status' column shows.

4. Schedule Calculator Application: If your task still doesn't run go to Control Panel->Scheduled Tasks. Click on Add Scheduled Task. From the list of applications, please select Calculator as shown below. If the calculator does not run then your scheduler service is not installed correctly on your computer.

My task is big. How do I test only a few commands in the task?

Select the command that you do not want to run. Right-click and select 'Disable'. Save the task and run it. while running the task it will not run the commands that are disabled. To enable the commands again right-click and select 'Enable'.

I want to copy a command and use it at many places. Do I need to create the command every time?

No. You can use the 'Duplicate' function by Right click on command and select 'Duplicate'. This command Duplicates the currently selected command. You can then just drag the newly created command using mouse.

I have created a keystroke that contains complex sequence of keys that I use repeatedly in many of my tasks. Do I need to copy the Keystroke every time I need to create a new task?

No. You can right click on that Keystroke and click on 'Convert To Variable'. You can then save the keystroke as a variable.

I have recorded a task that can be repeated as it is; only one text has to be replaced every time. I need to enter this text while running. How do I do that in your automation software?

1. Open the Task in the Editor

2. Right click on the Keystroke which you have to enter every time.

3. Click on the "Convert to Prompt" in the pop-up menu.

4. Enter the message that should be displayed which running.

5. Save the task.

6. While running, a dialog box will appear that will ask you to input the text that you wanted to enter.

7. Once you click ok then entered text will be typed.

Can I edit multiple commands at the same time? Is it possible to replace window title in all commands?

Yes. Bulk Edit is available for commands like keystrokes and delay. Select multiple commands of keystrokes or delay and right click on it. Select'Bulk Edit' option; specify the new details for the selected commands.

To replace the window title, select any command that has a window in it; right click and select 'Change Window Title' option. You can replace particular command's window title or you can change all commands windows title in a task.

What happens to a triggered or scheduled task if a high priority task is already running?

In the above case currently running high priority task will continue and all other task will be in queue.

Automation Anywhere features advanced task-queuing technology. This technology ensures that every task will complete based on its priority, regardless of how many other tasks interrupt the running task.

The order of execution is decided by the priority (high, medium and low). If two tasks in a queue have the same priority, the pre-cedence is decided by the category (triggered, scheduled and manual) of the tasks. Thus, if the two tasks have the same priority, a triggered task has higher precedence, followed by scheduled tasks, then manual tasks.

Can I use Automation Anywhere as a macro recorder?

Automation Anywhere can be used as an intelligent keyboard macro recorder and mouse macro recorder that allows you to easily create, edit and run macros.

- Automate keystrokes and save thousands of clicks to automate repetitive tasks.
- Record keyboard & mouse macros or use point & click wizards to create automated tasks in minutes.
- To learn more: http://www.automationanywhere.com/landing/macro-recorder.htm

Can I use Automation Anywhere to integrate Excel commands with other applications?

Automation Anywhere has an advanced Excel integration and Excel automation feature. Automation Anywhere allows automation of data transfer between Excel and any web or windows based application like Salesforce, SAP, Access, SQL, etc.

 Learn more on Excel Automation.

How does Automation Anywhere carry out database automation?

Automation Anywhere works with any SQL databases like Oracle, MS SQL, Sybase, etc. to transfer, maintain or update databases. Automation Anywhere's database automation solution can take data from various databases, web and in-house applications, collate them and generate reports.

Learn more on Database Automation.

Common Error Messages

This section provides you with information about common error messages that Automation Anywhere issues.

ERROR 429: ActiveX Component Can't Create Object

- **Description:** Error occurs due to problem in loading the User Interface of TA/AA
- **Solution:** 'Run Time Error 429' surfaces when the 'GDIPlusWrapper.dll' gets corrupted. You can type the below mentioned command under Start - Run - cmd:

```
For Windows 7 32 bit - regsvr32 C:\WINDOWS\system32\GDIPlusWrapper.dll

For Windows 7 64 bit - regsvr32 C:\WINDOWS\sysWOW64\GDIPlusWrapper.dll
```

Also See...

- Debugging Tasks
- Enabling the Debugging Option

Enabling the Debugging Option

Automation Anywhere provides a facility that enables you to debug your more complex automation tasks. Debugging tasks can be useful for automating longer, more complicated business or IT processes.

To enable the debugging option, follow these steps:

1. Open the task in the Task Editor.

2. Click on the Debug menu, and select Enable Debugging.

3. Use the toolbar to insert breakpoints in your task.

4. Debug the task line-by-line using the Step Over (F10) functionality.

You can use the Set SnapPoint feature in a task to capture images of the task while it runs.

Using the Debugging Option

You can use the Debugging option to do the following:

- To debug your task one command at a time, insert a breakpoint next to each command. This makes the task pause at the break-point.

- To insert a breakpoint, select a command and insert a breakpoint by clicking F9.

- To remove a breakpoint, select the command and click F9.

- Select the Step Over feature to move one command at a time.

- To clear all breakpoints, click Clear All Breakpoints.

- To stop debugging, click on the Debug menu and select Disable Debugging.

  > Note: Automation Anywhere does not save debugging information and breakpoints. When you close the Task Editor, all breakpoints are lost.

Also See...
- Using the Error View
- Viewing System Logs
- Logging into Windows when Application Paths Change

Frequently Asked Questions

- How does a user register as a Client with Automation Anywhere?
- Will a change in the domain name affect the working of my Client?
- Why is the client only able to view the 'Run' button?
- How do I copy an Excel cell and move to the next cell in the record?
- How do I paste data in an application and move to the next item?
- How do I create time stamps for files using Automation Anywhere system variables?
- What is the recommended way to open an application as part of a task?

How does a user register as a Client with Automation Anywhere?

1. Launch Automation Anywhere by clicking the Automation Anywhere Client icon.
2. In the Client Login window, click on the **Server Settings** link.
3. Enter the Server IP Address. If the client is on the same system as the server, enter 'localhost'.
4. Select the server Port used for the server.
5. Click on Apply and OK.
6. In the main client window, click on the Tools menu and select Server Communication.
7. Enter the user name and password provided while registering the client in the Automation AnywhereEnterprise Control Room.
8. Click the Login button.
9. If registered correctly, the message 'Registration Successful' is displayed.

Will a change in the domain name affect the working of my Client?

Yes. The Client that has already been registered and has a domain change, will have to be re-registered as the domain name is required especially while scheduling tasks and using the Auto-Login feature.

> Note You can verify the correct Domain Name in the Machine Information section of Control Room for that particular Client.

The Client that has had a domain name changed, can do any of the following (in the sequence mentioned):

1. Change the Application Path in Tools > Options > Advanced Settings. This will ensure that all the data that was available in the earlier path is available to the Client in the current domain.
2. Create a New Client. However, it is recommended that you use this option as a last resort.

Why is the client only able to view the 'Run' button?

If the Server Administrator does not grant permissions to a client, the client cannot view privileges other than Run.

1. To procure adequate permissions, request Create Task permissions from your Server Administrator.
2. The Administrator grants the Create Task privilege using the Enterprise Control Room, in the Client Control Center, using the Client Information section.
3. For Upload, Download, Delete, and View privileges for a particular folder on the server, request Access Control List permissions from the Enterprise Control Room.
4. After the Access Control List is updated in the Client Control Center and the Create Task privilege is granted, the client must re-login.
5. The client should now be able to communicate with the server using the new privileges.

For more information, see Using Client Control Center.

How do I copy an Excel cell and move to the next cell in the record?

When you highlight the Excel cell and use Ctrl+C to copy a cell, it copies an extra ENTER keystroke with the data. To copy only the cell, follow these steps:

1. Use the following keystrokes:

> **Keystrokes :**
> [F2] [HOME] [SHIFT DOWN] [END] [SHIFT UP] [CTRL DOWN] c [CTRL UP]

These keystrokes copy the cell data.

2. To move the cursor to the next cell after copying the current cell, use the following keystrokes:

- ENTER to move one row down in the same column
- TAB to move to the next cell to the RIGHT in the same row
- Shift + ENTER ([SHIFT DOWN][ENTER][SHIFT UP]) to move one row up in the same column
- Shift + TAB ([SHIFT DOWN][TAB][SHIFT UP]) to move to the next cell to the LEFT in the same row
- To skip a cell, use the corresponding key twice. For example, to move RIGHT to alternate cells, you must enter [TAB] [TAB]

How do I paste data in an application and move to the next item?

Most applications support the keyboard shortcut Ctrl+V to paste data that has been copied to the clipboard. To paste the data using the keyboard, follow these steps:

1. Ensure that the cursor is in the correct cell in a table, or in the correct control in a browser page, etc.

2. Type Ctrl+V. To simulate Ctrl+V in Automation Anywhere, use: [CTRL DOWN]v[CTRL UP].

   These keystrokes will paste the data in your application.

3. Use the following examples as a guide in how to move to the next record or cell in your application using the keyboard:

- Use the TAB key to move from a highlighted link to the next link in an Internet Explorer web page.
- Use the [RIGHT ARROW] to move to the next cell in the same row in a Microsoft Word table.
- Some applications also support using the Space bar to move to the next control or button.

How do I create time stamps for files using Automation Anywhere system variables?

Automation Anywhere provides the following system variables that you can use to append time stamps:

- Year
- Month
- Day
- Hour
- Minute
- Second
- Date

You can combine these variables to create a custom time stamp.

Examples:

- To append Year, Month, and Day at the end of ABC, you can use:

  `ABC$Year$$Month$$Day$`

- To create a complete time stamp with date and time, you can use:

  `ABC$Year$$Month$$Day$$Hour$$Minute$$Second$`

- You can also include text characters between the variables, for example:

  `ABC$Year$-$Month$-$Day$`

- To configure your own format for the Date variable, click on the Tools menu and select Variable Manager. Click on System Variable and then Date. If you use the Date variable to append a date to file names, be sure no slashes (/) exist in the date format, because file names in Windows might not use the / character.

What is the recommended way to open an application as part of a task?

The recommended methods for opening an application in a task include:

- When you record a task, double-click the application icon on the desktop.
- Click on the Start menu, and then Programs. Select the application.

If the location of these icons change, resulting in an error when the task runs, follow these recommendations:

1. Open the task in the Task Editor.
2. Select the actions that involve navigating to menu items and clicking. They might involve several mouse moves and mouse clicks.
3. Delete these actions.
4. Replace them with single-line commands that you create by double-clicking the Open Program/File command and specifying the path to the application. The Open Program/File dialog includes a text box in which you can type parameters that Automation Anywhere can pass to the application.
5. The task can now open your application, regardless of where the icon is located.

Logging into Windows when Application Paths Change
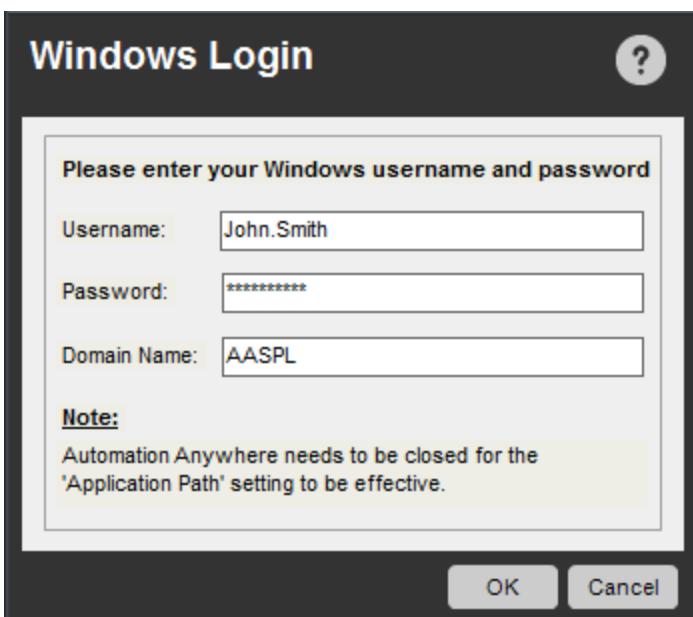
Occasionally, the application path for your Automation Anywhere tasks might require changing. For security reasons, you can set the Windows Login option in Advanced Settings to change the application path of your tasks.

Follow the steps given below:

1. Go to Tools -> Options -> Advanced Settings
2. Select the Application Path check-box if un-checked.
3. Click on the browse button

- The Windows Login command window is invoked:

4. Enter these parameters:

- **Username:** Enter the user name you created in User Accounts. How do I create a User Name and Password?
- **Password:** Enter the password that is associated with your user account.
- **Domain Name:** Specify the domain name. If not specified, the Windows Login uses the computer's domain name.

The new application path takes effect after you restart Automation Anywhere.

Object Cloning Command - Unable to find <object property name>

The Show Tip functionality of Automation Anywhere helps you easily understand and thereby rectify the error pertaining to object not found that is encountered when using the Object Cloning/SMART Recorder command.

On execution of a task, if an object is not found you will be shown an error that takes into consideration the various parameters that have been input.

To view the message click on 'Show Tip' link. This will display a possible solution. If however, this does not suffice, you can click on the 'Learn more' link which will guide you to this topic.

Tips displayed

While using wild-card when the Search criteria do not match

1. Click on Show Tip link on the Error Message.
2. The Error messages clearly indicates the object not found with recommended action.
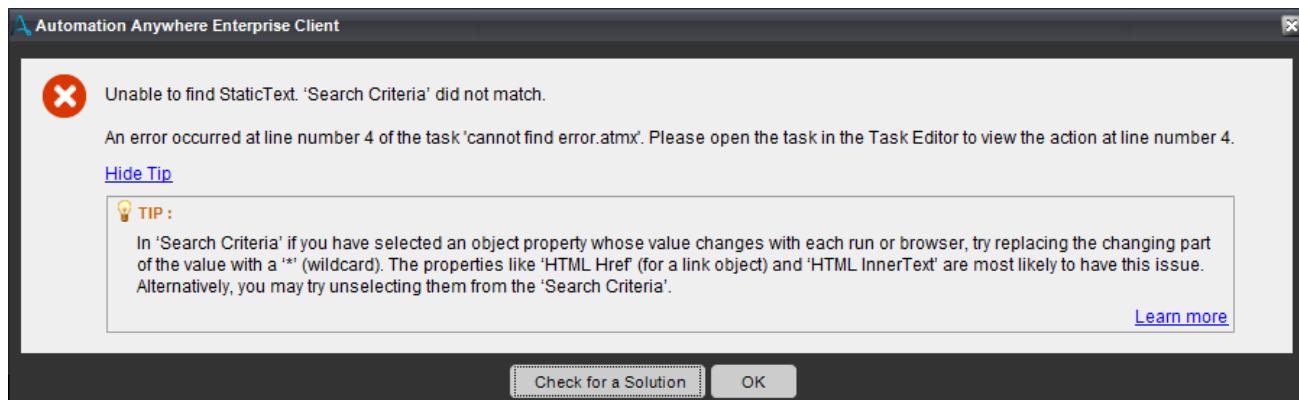3. The user can try modifying the Search Criteria with Wild Card.
4. Use * for specific properties.
5. One can use * at beginning, end or middle for any property.
6. One can also view the same tip details in Test Log as shown below.

<content missing>

More Tips as per Technology when Object not found

- **MSAA:** If object not found and HTML Properties are checked in Search Criteria then Provide wildcard support in value or Uncheck HTML properties like HTML Href, HTML InnerText.

  If object not found and HTML Properties are not checked in Search Criteria then check other Unique properties like Name, Value, Parent, Index
- **Silverlight:** If object not found then check other Unique properties like Name, Value, Parent, Content
- **Flex:** If object not found then check other Unique properties like Name, Value, ObjectID, Parent
- **UI Automation:**
  - **Java:** If object not found then check other Unique properties like Name, Value, ObjectID, Parent, Index
  - **HTML:** If object is not found then check other Unique properties like HTML Tag, HTML Tag-Index, HTML Name, HTML ID.

If object not found and HTML Properties are checked in Search Criteria then Provide wild-card support in value or Un-check HTML properties like HTML Href, HTML InnerText

When to modify Search Criteria when objects do not have common/default properties

1. Click on Show Tip link on the Error Message.
2. The Error messages clearly indicates the object not found with recommended action.
3. Modify the search criteria to include unique object properties for the test to execute successfully.

4. If object property is dynamically changes during run time, try un-checking properties, and the object will be found.

5. One can also view the same tip details in Test Log as shown below.

<content missing>

Below table displays the main search criteria for different objects with respect to various technologies.

		Textbox/Static Text	Link/Image	Other
1.	Common to All	Type Path	Type Path	Type Path
2.	.NET/WPF	Name Object ID	Name Object ID	Name Object ID
3.	Silverlight	Parent Name	Parent Name	Parent Name
4.	Flex	Class Index	Class Index	Class Index
5.	MSAA	Name	Name	Name
6.	MSAA-HTML	HTML ID HTML Name HTML Inner Text	HTML Href HTML Name HTML Inner Text	HTML ID HTML Name
7.	HTML	HTML Value HTML Tag HTML Type HTML Title HTML Name HTML ID HTML TagIndex HTML Frame Src HTML Frame Name HTML Frame Path	HTML Href HTML Inner Text HTML Value HTML Tag HTML Type HTML Title HTML Name HTML ID HTML TagIndex HTML Frame Src HTML Frame Name HTML Frame Path	HTML Value HTML Tag HTML Type HTML Title HTML Name HTML ID HTML TagIndex HTML Frame Src HTML Frame Name HTML Frame Path

Troubleshooting Information

These topics provide troubleshooting information to help you diagnose and fix problems that might arise while installing, configuring, and using an Automation Anywhere client.

Information pertaining to the following has been described:

- Troubleshooting Installation Issues
- Running a Task - Troubleshooting
- Scheduling - Troubleshooting
- Auto-Login Issues
- Triggers not working correctly with Microsoft Office files
- Capture button disabled in Web Recorder despite clicking Launch

Troubleshooting Installation Issues

1.Error while unpacking program, code LP5

If you receive an error while unpacking the program code, you might need to add Automation Anywhere to the firewall exceptions list.

Follow these steps to add Automation Anywhere to the exceptions list:

1. Open the Windows Control Panel (all items view).
2. Click on the Windows Firewall icon.
3. Click on the "Allow a program through Windows Firewall" link in the left blue pane.
4. Click on the Change Settings button.
5. Allow an existing exception by completing these steps:
   a. On the left side, check the program exception name box to allow the program through Windows Firewall.
   b. Check (allow) the box for each Private or Public network location for which you want the program to have access through Windows Firewall.
6. Add a program exception by completing the following steps:
   a. Click on the "Allow another program" button.
   b. If the Automation Anywhere program is listed, click on the program to highlight it.
   c. If the Automation Anywhere program is not listed, click on the Browse button and navigate to: "C:\Program Files\Automation Anywhere\Automation Anywhere.exe" This is the Automation Anywhere executable file location.
   d. Click the Open button.
   e. Click the Add button.
7. When done, click OK.
8. Close the Windows Firewall window.

2. MSSTDFMT.DLL Error Windows 7 only

If you receive this error message, follow these steps to resolve it:

1. Download the file 'mssfdfmt.zip' from the location:
2. http://www.dll-files.com/dllindex/dll-files.shtml?msstdfmt
3. Save this file to any location on your system and extract it.
4. Copy the MSSTDFMT.DLL file from the downloaded folder and paste it to this folder location: 'C:\Windows\syswow64\sysWOW64'
5. In the folder 'sysWoW64', locate the file 'cmd.exe'.
6. Right click on 'cmd.exe' and click on 'Run as administrator'. A Windows command prompt window is displayed showing 'C:\Windows\system32>'
7. Type: Regsvr32 C:\Windows\SysWOW64\MSSTDFMT.DLL and press Enter to register the file.

Completing these steps should resolve the error.

Running a Task - Troubleshooting

A task that contains Microsoft Excel commands runs very slowly. In this case, follow these steps:

1. Disable Debug mode.

2. Remove or reduce Delay commands by 1 second.

3. Minimize the Excel files. While Excel is in a maximized state, it displays the user interface, consuming resources.

4. Stop capturing screen shots while running the task. Follow these steps:

    a. On the Automation Anywhere main window, click on the Tools menu and select Options.

    b. Select Run Time Settings.

    c. Uncheck the option: "Capture Screenshots while Running a Task".

Be aware that screenshots will not be captured while running the task, so monitoring progress is more difficult.

Scheduling - Troubleshooting

1. Scheduled Task Not Running

If a scheduled task fails to run using the scheduler, follow these steps to resolve the problem:

1. Ensure that you specify the correct Windows user name and password.

    - For Windows XP: For security reasons, Windows XP requires that your Windows XP user name and password be specified to run a scheduled task. For more information, see "Scheduling a Task".

    - For other Windows Operating Systems: If your account requires a password to log on, specify that password when scheduling tasks. Otherwise, no password is required.

2. If you have multiple accounts, ensure that you have provided the correct user name and password. If you have multiple accounts on the same computer (such as a domain account and a local account), verify that the user name provided is the same as that used to log onto the computer.

3. Scheduled tasks do not run if you schedule them using a user account different from that logged on. Automation Anywhere automatically fills in the user name that was used to log in.

4. Check the status in Control Panel by completing the following steps:

    a. Open the Windows Control Panel and go to Administrative Tools > Task Scheduler.

    b. Select the task corresponding to your task (the name begins with your task name) and right-click on it.

    c. Click on Run to verify that the task runs.

    - If the task does not run, check the Status column and go to Step

> Note: The above steps are valid only for Automation Anywhere Small Business version.

Schedule the Calculator Application by completing these steps:

1. If your task still fails to run, open the Windows Control Panel and click on Scheduled Tasks.

2. Click on Add Scheduled Task.

3. From the list of applications, select Calculator (see screen-shot below).

    - If the calculator fails to run, the scheduler service is not installed correctly on your computer. See your System Administrator.

- Ensure that the Task Scheduler Service is started. By default, the Scheduler service is started on all Windows XP machines.

To verify that it is running, complete the following steps:

1. Click on the Windows Control Panel and select Administrative Tools and then Services.
2. Verify that the status of the Task Scheduler service reads 'Started.'
3. If the Task Scheduler is not running, right-click on it and select Start. Also, ensure that the 'Startup type' is set to 'Automatic'. This starts the service automatically the next time you start your system.

- Restart the Task Scheduler.

If the Task Scheduler still fails to run, complete the following steps:

1. Click on the Windows Control Panel and select Scheduled Tasks.
2. In the Advanced Menu, click 'Stop using Task Scheduler.'
3. Wait 10 minutes.
4. Click on 'Start using Task Scheduler.'

2. Migrating schedules to a newer version

Applicable from version 8.0 onward.

If using a version prior to 8.0, you are required to migrate schedules in the Client from earlier versions to the latest one.

A schedule migration utility, bundled within the setup, is available post installation. Run this utility to save time on automating path updates, which are now required to run the older schedules, successfully.

All you need to do is:

1. Run the migration utility without Admin privileges to migrate all the schedules by the logged in user. A logged in user need only provide his/her Windows credentials
2. Run migration utility with Admin privileges to migrate all the schedules pushed by the Control Room admin. User will need to provide their Windows credentials to complete this activity.

📝 Note: After migration, all schedules of earlier version, pushed by the Control Room admin will be visible to the user. However, the new schedules will continue to remain invisible on the Client side.

Schedule migration utility path:

<AA Installation Path>\AutomationScheduleMigration.exe

For instance:

> C:\Program Files (x86)\Automation Anywhere Enterprise 8.0\Client\AutomationScheduleMigration.exe

[Back to the top](#)

Auto-Login Issues

You have issue with Auto-Login? Check the below:

- Ensure that SAS.dll is present in system32 (e.g. C:\Windows\System32) folder in case of Vista OS and above. If it is not there, then copy the same from the product installation folder (e.g. C:\Program Files\Automation Anywhere 7.1\Client\32-Bit OR from 64-Bit; depending on OS)
- Verify for proper working of Ctr+Alt+Del:
- Registry: HKEY_LOCAL_MACHINE\SOFTWARE\Microsoft\Windows\CurrentVersion\Policies\System\SoftwareSASGeneration
- Value: Dword 3
- Verify Auto-Login Registry and value.

    [HKEY_CLASSES_ROOT\CLSID\{75A22DF0-B81D-46ed-B119-CD30507BD614}]@="Automation.CredentialProvider"

    [HKEY_CLASSES_ROOT\CLSID\{75A22DF0-B81D-46ed-B119-CD30507BD614}\InprocServer32]@="Aut-tomation.CredentialProvider.dll""ThreadingModel"="Apartment"

- Verify Auto-Login service is enabled and running.
- Goto services.msc -> 'Automation Anywhere Enterprise Auto Login Service'

- Credentials cross-check:
    1. Select Task > Provide schedule > Enter window authentication credentials with domain e.g. User name :domain\username & Password
    2. If the schedule has been successfully added, go to Tools > Option > Auto-login Settings. Check the username and password.
- It should match.

Back to the top

Triggers not working correctly with Microsoft Office files

To use a trigger for Microsoft Office file when a file is modified, we need to use the condition for File -> When file is renamed.

The trigger will be fired correctly for modified as well as renamed conditions. This is the inherent behavior for Microsoft Office.

Back to the top

Capture button disabled for options 'Extract Source' and 'Execute Java Script' in Web Recorder command

The 'Capture' button will get enabled for options:

- **Extract Source -** if the page that is launched has an iFrame.
- **Execute Java Script -** if the page that is launched has a java-script.

Back to the top

Contact Technical Support

After completing the previous steps, if you are unable to run your task using the Task Scheduler, it is likely that the Scheduler service is not installed correctly on your computer. Please contact Automation Anywhere Technical Support for further assistance.

Using the Error View

The Automation Anywhere Task Editor provides two options for viewing and fixing task errors and working with task variables. To activate the Error View and the Variable Manager, click on these links:

Error View

Use the Error View to manage errors that occur in your tasks. This view is automatically updated when the Error Handling feature is used.

Variable Manager

Use the Variable Manager to create and manage variables that you use in your tasks. You can add, edit, or delete local (user-defined) variables.

Variable Watch Table

Use the Variable Watch Table to track the values of your variables. The Variable Watch Table is enabled when you enable Debug Mode by clicking on the Enable Debugging link.

You can move the Variable Watch Table, as well as expand it as required.

To view how variables dynamically change, click the Add or Remove buttons. You can view both user-defined variables and system variables.

Use the Variable Watch Table as part of your debugging process.

Also See...

- Enabling the Debugging Option
- Viewing System Logs
- Logging into Windows when Application Paths Change