

Unattended Testing How To

You can use QA Wizard Pro to shorten your testing cycle by combining scripts into a single file that runs unattended overnight or longer. This How To walks you through creating, configuring, scheduling, and running a QA Wizard Pro batch file.

Unattended Testing Overview

You can test your application unattended by creating and running a batch file in QA Wizard Pro that includes your testing scripts and any associated datasheets.

By running a batch file of your testing scripts, you can:

- Run your testing scripts for a set amount of time
- Run your scripts unattended
- Run scripts from multiple computers
- Regression test your application as part of a nightly build process
- Schedule scripts to run during off hours
- Save time and complete your testing faster

Unattended Testing Scenario

This unattended testing scenario describes how to create, configure, and run a QA Wizard Pro batch file that executes your testing scripts unattended.

Setup

Before you begin this unattended testing scenario, take the time to perform the following setup tasks:

- **Set up the testing environment**—Make sure you have access to the hardware, software, and network resources you need to record or run tests.
- **Create a workspace**—Workspaces organize scripts and related reports and datasheets. Depending on your organization's process, you may use one workspace for each application or one workspace for each functional area.
- **Set up the application in the application repository**—Application repositories store information about the tested application and version. Each version contains window and control data that identifies and locates objects.
- **Set general and playback options**—Take a few minutes to set general options that control how QA Wizard Pro works and set playback options that control how scripts run.
- **Record or update scripts**—Make sure you have recorded or updated all the scripts that you want to run unattended.

Unattended Testing Instructions

Before you create the batch files to run your automated testing scripts unattended, you must first record or update your scripts.

Note: Try to keep your testing scripts short and task-based. This makes them easier to manage and maintain. Tests should be repeatable and have an expected result.

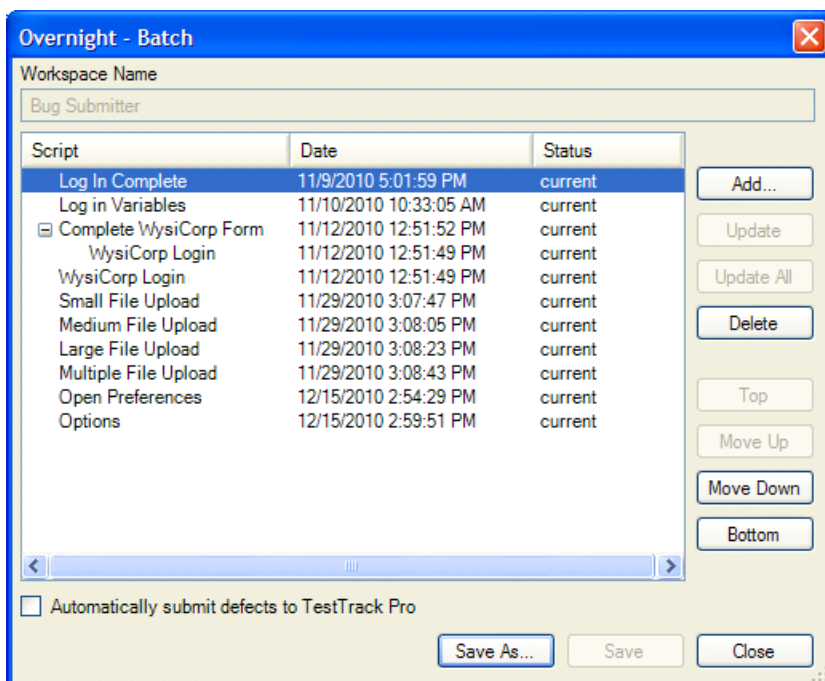
Creating batch files

After you record and update your test scripts, you can create the batch file that runs your unattended scripts.

Note: Batch files can only include scripts and datasheets from the same workspace.

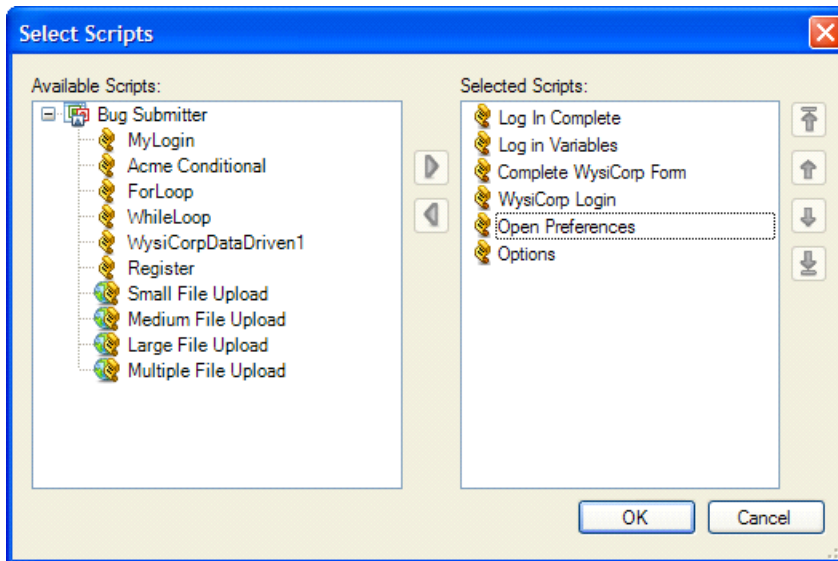
1. Choose **Batch > New Batch**.

The Batch dialog box opens.



2. Click **Add**.

The Select Scripts dialog box opens.



3. Select the scripts you want to add to the batch file and click the right arrow button to add them to the Selected Scripts list.
4. Select a script and click **Top**, **Move Up**, **Move Down**, or **Bottom** to reorder the list.
Scripts run in the order displayed.
5. Click **OK**.
6. Optionally select **Automatically submit defects to TestTrack Pro** to add a defect for each failed script to TestTrack Pro.
7. Click **Save As**.
The Save As dialog box opens.
8. Enter a **File name**, select a **Save in** location or browse to select a different location, and click **Save**.
Batch files are saved with a qawbatch file extension.
9. Click **Close** to close the Batch dialog box.

Note: When a script has a CallScript statement or a datasheet associated with it, QA Wizard Pro automatically includes the called script or datasheet in the batch. The script name in the CallScript statement must be a text string. If OpenRecordset functions are used to access local datasheets, the datasheet name must be a text string. Variables and other expressions are not supported.

Modifying batch files

When you create a batch file, QA Wizard Pro makes a copy of the scripts and the associated datasheets and bundles them in the batch file. You can modify a batch file to change the order that scripts run or remove scripts and datasheets.

1. Choose **Batch > Open Existing Batch**.
The Open dialog box opens.
2. Select the batch file you want to modify and click **Open**.

The Batch dialog box opens. Information about each file is displayed in columns.

| Column Name | Description |
|-------------|--|
| Script | Script file name |
| Date | Date the script or datasheet was last updated in the batch file |
| Status | Status of the script or datasheet in the batch file: <ul style="list-style-type: none"> ■ Current—File in the batch matches the file in the workspace ■ Old—File in the workspace has changed ■ Missing—File cannot be found in the workspace ■ Unknown—The workspace that contains the file is not open |

Note: Called scripts and datasheets are displayed under the related script.

3. Make any of the following changes to the batch file.
 - Select a script or datasheet with an Old status and click **Update** to replace it with the updated file.
 - Click **Update All** to update all files. You cannot update files with a Missing or Unknown status.
 - Select a script and click **Top**, **Move**, **Move Down**, or **Bottom** to reorder the list.
 - Select a script and click **Delete** to remove it from the batch file. You cannot delete called scripts or associated datasheets.
4. Click **Save** to save the changes.
5. Click **Close** to close the Batch dialog box.

Running batch files

You can run QA Wizard Pro batch files immediately or schedule them to run unattended. Keep the following in mind:

- If you move a batch file to another computer to run it, the applications must be available in the global or local application repository.
- When you run batch files, the playback options set on the computer running the scripts are used.
- If the batch file includes web scripts, the selected browser runs all scripts in the batch.
- Batch file run reports are stored in the Reports folder in the workspace directory (e.g., C:\Documents and Settings\\My Documents\QA Wizard Pro Workspaces\Workspace\Reports).

Running batch files from Windows Explorer

You can double-click a batch file from Windows Explorer to run it immediately.

1. Navigate to the location of the batch file (for example, C:\Documents and Settings\\My Documents\QA Wizard Pro\Batch Files\Overnight.qawbatch).
2. Double-click the batch file you want to run.

A command prompt opens and the batch file runs.

Running batch files from a command prompt

You can use the QAWRunScript command to run batch files from a command prompt.

1. Choose **Start > Run** and enter `cmd` in the **Open** field.
2. Click **OK**.
A command prompt opens.
3. Enter `cd\program files\seapine\QA Wizard Pro` to navigate to the QA Wizard Pro directory.
4. Enter `QAWRunScript.exe <name of batch file>.qawbatch` to run the batch file.

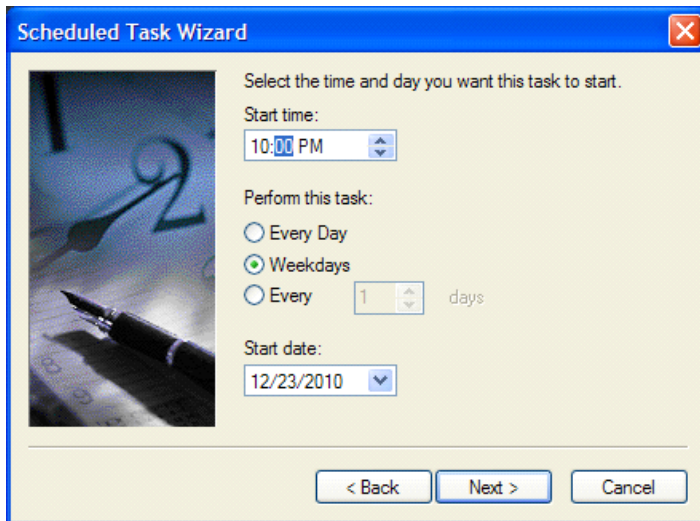
Scheduling batch files to run unattended

You can use the Windows Scheduled Task Wizard to configure your test machine to run the batch file unattended at the date, time, and interval that works best for you.

1. Choose **Start > Programs > Accessories > System Tools > Scheduled Tasks**.
2. Double-click **Add Scheduled Task**.
The Scheduled Task Wizard opens.
3. Click **Next**.
4. Click **Browse** and navigate to the batch file you want to schedule.
5. Select the batch file and click **Open**.
6. Enter a name for the task to run the batch file, select the interval (Daily, Weekly, etc.), and click **Next**.



7. Enter the **Start time**, select the days you want to perform the task, and enter the **Start date**.



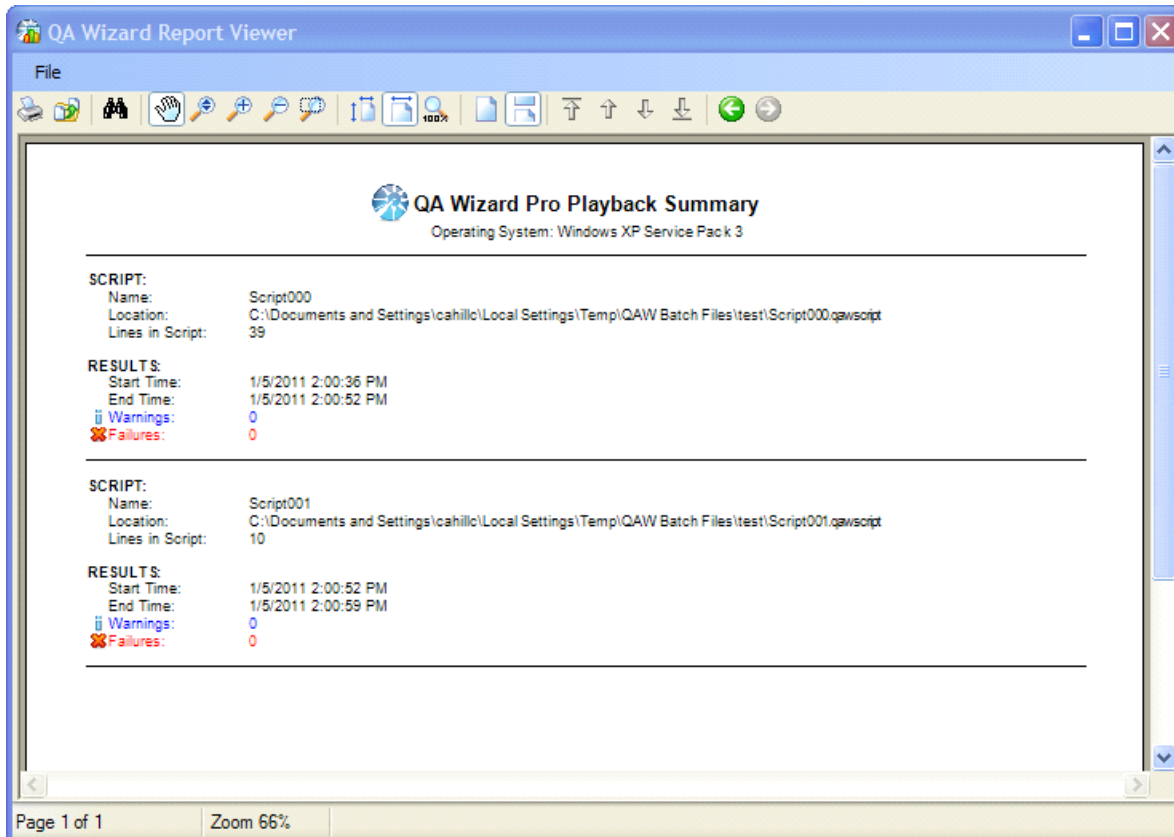
8. Enter the username and password of the user account you want Windows to use to run the batch file.
9. Click **Open advanced properties for this task when I click Finish** if you want to set advanced configuration options after you click Finish.
10. Click **Finish** to schedule the batch file to run.

Tip: All scheduled tasks are stored in the C:\Windows\Tasks directory.

Unattended Testing Reports

QA Wizard Pro generates and saves a report when your batch file finishes running. The report includes information about all the scripts included in the batch file, such as the test computer configuration, the script name and location, a summary of the test results, and list of warnings if any were encountered. Reports are named using the name of the first script in the batch file and the date and time the script ran.

Batch file reports are stored in the Reports folder in the workspace directory (for example, C:\Documents and Settings\\My Documents\QA Wizard Pro Workspaces\\Reports). You can view the report by double-clicking on it.



Unattended Testing Conclusions

Creating a batch file allows you to save time, run more testing scripts, and complete them much faster than you would running each one individually. Using a combination of individual testing scripts and batch files scheduled to run during off hours, you can thoroughly test your application and provide a more stable product to your users.

Links to other resources

You can view the following resources for more information on QA Wizard Pro.

- [QA Wizard Pro Resource Center](http://www.seapine.com/qawevaltools.php) - <http://www.seapine.com/qawevaltools.php>
- [QA Wizard Pro Blog](http://blogs.seapine.com/category/products/qawizardpro/) - <http://blogs.seapine.com/category/products/qawizardpro/>
- [Knowledgebase](http://www.seapine.com/kb/categories/QA+Wizard+Pro/) - <http://www.seapine.com/kb/categories/QA+Wizard+Pro/>