

Getting Started with Your QA Wizard Pro Load Test Evaluation

Welcome to QA Wizard Pro! This guide walks you through the basic steps to record and run a script that load tests your web application.

Need help?

Your evaluation includes free technical and customer support. If you have any pre-sales, installation, or evaluation questions please email support@qawizard.com or visit www.qawizard.com/support for more information.

Step 1: Before you start

Keep the following in mind during your evaluation:

- If you have a QA Wizard Pro license, you can record and run load test scripts without requesting additional licenses. You are limited to five virtual users regardless of the number of copies of QA Wizard Pro you install on your network until you purchase non-evaluation virtual users.
- Load test scripts do not capture user interface actions like standard QA Wizard Pro scripts. When you record a load test script, only information exchanged between your application and the web server hosting it is captured and added to the script.

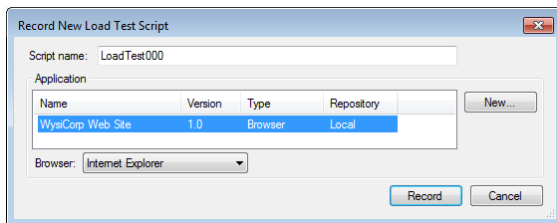
Step 2: Create a workspace

Before you start recording load test scripts, you need to create a workspace to organize scripts and reports.

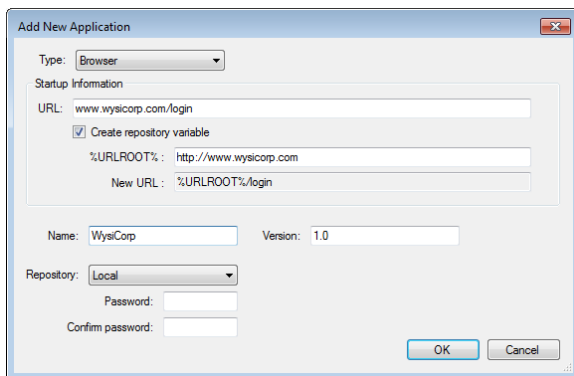
1. Choose File > New > New Workspace. The Add New Workspace dialog box opens.
2. Enter a workspace File name. Select a Save in location or browse to select a different location.
3. Click **Save**. The workspace is created. You are now ready to record a script.

Step 3: Record a load test script

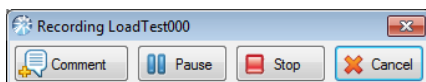
1. Choose Script > Record Load Test. The Record New Load Test Script dialog box opens.



2. Enter a Script name.
3. Click **New** to add the application you are testing. The Add New Application dialog box opens.



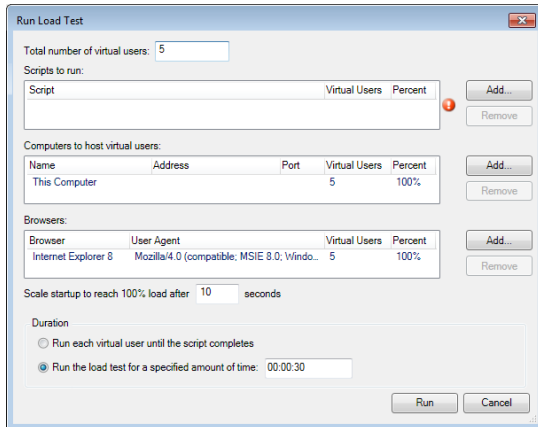
4. Select **Browser** and enter the application Startup information and Name.
5. Click **OK** to save the changes. You return to the Record New Load Test Script dialog box.
6. Select the Browser to run the application in.
7. Click **Record**. The application starts and the Recording toolbar opens.



8. Navigate through the application, performing actions you want to test with a simulated load of users.
9. Click the **Stop** Recording toolbar button when you finish recording. The script is created and opens in the Script pane.

Step 4: Run the load test script

1. Choose Script > Run Load Test. The Run Load Test dialog box opens.

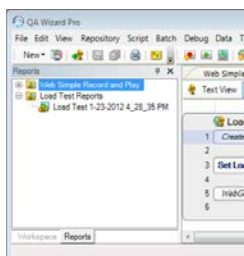


2. Accept the default value (5) in the Total number of virtual users field. Your evaluation license includes five virtual users.
3. In the Scripts to run area, click **Add**, select the script you recorded, click the arrow to add it to the Selected Scripts list, and click **Select**. The script is added to the Scripts to run list.
4. Skip the Computers to host virtual users area. For your evaluation, all virtual users are hosted on the current computer. If you purchase additional virtual user licenses, you can host additional virtual users on remote computers and specify the number of users to simulate on each computer.
5. In the Browsers area, click **Add** to use additional browsers or change the default browser for the test. If you use multiple browsers, enter the number of Virtual Users or the Percent of virtual users to simulate using each browser. The sum of virtual users must be 5 because that is the number of total users specified for the test.
6. Skip the Scale startup to reach 100% load field, which is used to specify the number of seconds spent loading virtual users.
7. Select a test Duration. Select Run each virtual user until the script completes to stop the test when the script is complete. Select Run the load test for a specified amount of time to stop the test when the specified time limit is reached. If the script stops before time expires, it restarts and continues to run for the duration of the specified time.
8. Click **Run**. The load test runs. A progress indicator opens and displays the run time, current load, and number of errors and warnings returned during the test.

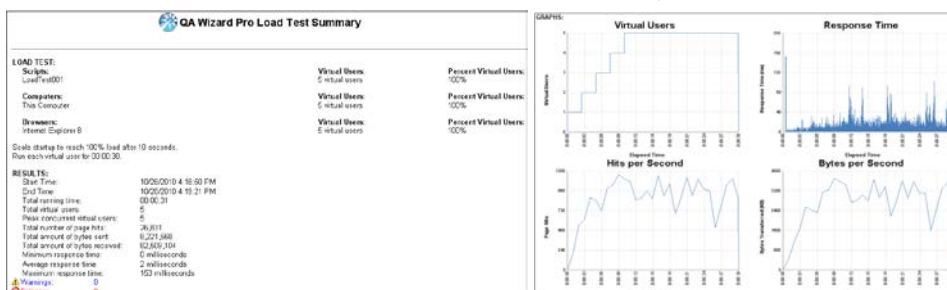
Step 5: View the load test report

When the test is complete, a results report is generated. The report includes information about how well the server performed under load and includes graphs that you can export.

1. Click the Reports tab in the Workspace pane.
2. Expand the Load Test Reports folder.



3. Double-click the report that corresponds to the date and time you ran the test. The report opens.



Need more help getting started? See 'Load Testing' in the [QA Wizard Pro User Guide](#).