

IBM® Rational® Test Automation Server
10.5.1 Documentation
December 2022

Special notice

Before using this information and the product it supports, read the information in [Notices on page dclxxv](#).

Contents

Chapter 1. Release Notes.....	8
Product description.....	8
What's new.....	8
Installing and upgrading.....	10
Known issues.....	10
Contacting IBM Rational Software Support.....	10
Chapter 2. System Requirements.....	12
Hardware.....	13
Operating systems and Containers.....	13
Host prerequisites.....	15
Supported software.....	16
Chapter 3. Getting Started Guide	19
Server overview.....	19
Supported versions of assets and resources.....	20
Suites and tests supported.....	21
Virtual services supported.....	23
Task flow: Test runs and results.....	24
Task flow: Server access from desktop clients.....	26
Accessibility features.....	28
Chapter 4. Administrator Guide.....	29
Installation of the server software.....	29
Installation of the server software on IBM® Red Hat OpenShift.....	30
Installation of the server software on Ubuntu.....	47
Installation of the server software on Azure Kubernetes Service.....	57
Management of server features.....	68
Server software upgrade methods.....	71
Server software upgrade on Red Hat OpenShift.....	72
Server software upgrade on Ubuntu.....	77
Upgrading the server software on Azure Kubernetes Service.....	79
Migration of user data.....	82
Migrating user data on Red Hat OpenShift.....	85
Migrating user data on Ubuntu from 10.1.2 or earlier.....	86
Migrating user data on Ubuntu from 10.1.3 or later.....	88
Backup and restoration of the server data.....	89
Backup and restoration of the server data on IBM® Red Hat OpenShift.....	90
Backup and restoration of the server data on Ubuntu.....	93
Backup and restoration of the server data on Azure Kubernetes Service.....	97
Uninstallation of the server software.....	102
Uninstalling the server software from IBM® Red Hat OpenShift.....	102
Uninstalling the server software from Ubuntu.....	103
Uninstalling the server software from Azure Kubernetes Service.....	104
Configuration of the server software.....	105
User administration.....	106
Certificate authority: Importing and extending lists.....	115
Copying third-party application Jars to Kubernetes.....	118

Changing the password seed.....	121
Server extensions.....	123
Team space administration.....	125
Team space overview.....	126
Creating a team space.....	129
Tasks in a team space.....	131
Tasks in projects in a team space.....	134
Management of licenses in Rational® Test Automation Server.....	136
Chapter 5. Test Author Guide.....	146
Edit branch overview.....	146
Datasets overview.....	147
Creating a dataset.....	148
Creating a dataset by using a CSV file.....	150
Editing a dataset.....	152
Publishing a dataset.....	162
Viewing a dataset.....	163
Deleting a dataset.....	164
Chapter 6. Test Execution Specialist Guide.....	167
System modeling.....	167
Task flows for working with a system model.....	171
Adding a repository to a team space.....	173
Creating a system model.....	175
Creating components.....	177
Creating child components.....	178
Creating linkages between components.....	180
Deleting linkages between components.....	182
Modifying components.....	184
Deleting components.....	186
Publishing changes to the system model.....	187
Associating resources with components.....	187
Viewing resources that are associated with components.....	189
Removing resources that are associated with components.....	193
Publishing changes of associating or dissociating resources with components.....	195
Viewing the system model.....	197
Deleting a system model.....	197
Deleting a repository that is added to a team space.....	198
Prerequisites to running tests.....	198
Test run considerations for AFT Suites or Functional tests.....	199
Test run considerations for API Suites or API tests.....	200
Test run considerations for schedules.....	203
Considerations for using Jaeger traces in reports.....	205
Test run considerations for JMeter tests.....	205
Test run considerations for JUnit tests.....	207
Test run considerations for Postman tests.....	208
Management of agents.....	210
Management of performance agents.....	210
Management of Docker hosts.....	218

Test run considerations for running tests on remote Docker hosts.....	218
Setting up a remote Docker host.....	220
Setting up a secure remote Docker host.....	222
Copying third-party application Jars to a remote Docker host.....	223
Registering a remote Docker host.....	225
Viewing remote Docker hosts that are registered with IBM® Rational® Test Automation Server.....	227
Adding a remote Docker host.....	228
Editing configurations.....	229
Deleting a remote Docker.....	232
Unregistering a remote Docker host from IBM® Rational® Test Automation Server.....	233
Management of remote Kubernetes clusters.....	234
Considerations for using a remote Kubernetes cluster.....	235
Registering a remote Kubernetes cluster.....	237
Viewing remote Kubernetes clusters.....	240
Adding a remote Kubernetes cluster to a project.....	242
Editing configurations of a remote Kubernetes cluster.....	244
Removing a remote Kubernetes cluster from a project.....	246
Unregistering a remote Kubernetes cluster from a team space.....	248
Test run configurations.....	249
Configuring a Suite or test for a quick run.....	250
Configuring an AFT Suite run.....	255
Configuring an API Suite or an API test run.....	267
Configuring a run of a Compound Test that contains HTML tests.....	280
Configuring a run of a Compound Test that contains Web UI tests or a Functional Web UI test	291
Configuring a run of a Compound Test that contains mobile tests or a Functional Mobile test.....	304
Configuring a run of a Functional Windows test.....	319
Configuring a run of a Compound Test that contains performance tests or a Performance test.....	331
Configuring a run of a JMeter test.....	344
Configuring a run of a JUnit test.....	353
Configuring a run of a Postman test.....	362
Configuring a run of a Rate Schedule or VU Schedule.....	371
Configuration of a scan of an application code.....	383
Considerations to scan an application code.....	384
Configuring a run to scan the application code.....	386
Management of running tests.....	394
Viewing the state of test assets.....	394
Viewing the progress of running test assets.....	398
Checking logs.....	401
Resetting the configuration settings for a test run.....	402
Stopping test runs.....	403
Canceling scheduled test runs.....	406
Management of virtualized services.....	408
Prerequisites for running virtual services.....	410
Viewing intercepts that are registered with a team space on IBM® Rational® Test Automation Server.....	415
Viewing virtual service resources.....	416
Configuring a run of a virtual service.....	419
Running HTTP virtual services without using proxies.....	433

Viewing running instances of virtual services.....	438
Viewing configurations of running instances of virtual services.....	443
Modifying configurations of running instances of virtual services.....	447
Viewing routing rules of the virtual services.....	449
Viewing usage statistics of virtual services.....	451
Stopping virtual services.....	454
Test results.....	455
Test results and reports.....	456
Test results data in Grafana.....	465
Viewing Surefire format reports.....	469
Test logs.....	470
Test log queries	484
Resource monitoring.....	487
Resource monitoring capabilities.....	489
Resource monitoring agents.....	507
Managing sources.....	519
Collecting and viewing resource monitoring usage data.....	526
Configuration of a change management system.....	532
Configuration of Atlassian Jira as a change management system.....	533
Integrations with other applications.....	543
Integration plug-in compatibility matrix.....	543
Integration with Azure DevOps.....	544
Integration with IBM® Engineering Test Management.....	555
Integration with Jenkins.....	562
Integration with UrbanCode™ Deploy.....	577
Integration with Xray test management for Jira.....	593
Managing access to IBM® Rational® Test Automation Server.....	600
Working with IBM® Rational® Test Automation Server REST services.....	601
Chapter 7. Test Manager Guide.....	603
Managing team spaces.....	603
Viewing team spaces.....	603
Adding members to a team space.....	604
Modifying the configuration of a team space	607
Viewing the configuration of a team space	609
Becoming a team space member.....	611
Managing members and their roles in a team space.....	612
Deleting a team space.....	615
Test assets and a server project.....	616
Working with projects in a team space.....	616
Repository considerations for a server project.....	617
Adding a project.....	619
Adding repositories to a server project.....	620
Secrets configuration	623
Adding users to a project.....	623
Viewing active projects.....	624
Changing the visibility configuration of a project.....	625
Becoming a project member.....	626

Managing access to server projects.....	627
Archiving or unarchiving server projects.....	629
Deleting server projects.....	630
Project overview.....	630
Managing repositories.....	632
Adding repositories to a server project.....	632
Updating the authentication credentials of the repository.....	635
Deleting a repository.....	636
Refreshing repositories manually.....	636
Selecting the global branch in a project.....	638
Creating webhooks.....	639
Protecting API test assets by using secrets.....	640
Managing secrets collections.....	640
Granting access to members or member roles.....	643
Creating a secret in a secrets collection.....	642
Managing secrets.....	645
Managing an encrypted dataset.....	646
Creating a classification.....	647
Editing or deleting a classification.....	648
Moving an encrypted dataset to another classification.....	649
Removing a dataset from the classification.....	650
Granting classification access to members or members roles.....	651
Managing notifications.....	652
Managing in-app notifications.....	653
Managing email notifications.....	658
Chapter 8. Troubleshooting.....	663
Troubleshooting issues.....	663
Security Considerations.....	dclxxiii
Notices.....	dclxxv
Index.....	679

Chapter 1. Release notes for IBM® Rational® Test Automation Server

This document includes information about What's new, installation and upgrade instructions, known issues, and contact information of IBM Customer Support.

Contents

- [Product description on page 8](#)
- [What's new on page 8](#)
- [Installing and upgrading on page 10](#)
- [Known issues on page 10](#)
- [Contacting IBM Rational Software Support on page 10](#)

Product description

You can find the description of IBM® Rational® Test Automation Server.

Rational® Test Automation Server is a server that includes capabilities such as project and role-based security, Docker-based distribution and installation, and running of test cases. For more information about the server, see [IBM Rational Test Automation Server overview on page 19](#).

What's new

You can find information about the features introduced in this release of Rational® Test Automation Server.

The following sections list the new features, enhancements or other changes made in this release.

- **Support to export test results to a Jira Xray project**

When you use Atlassian Jira as a change management system on Rational® Test Automation Server and you use Xray as the test management application in Jira, you can now integrate Xray with Rational® Test Automation Server. With this integration, you can export the test results of the tests that run on Rational® Test Automation Server to an issue in your Jira Xray project. You can then view and use the test results in your project in Jira.

See [Integration with Xray test management for Jira on page 593](#).

- **Configuring Azure DevOps as a change management system**

You can now add Azure DevOps as a change management system on Rational® Test Automation Server, if you use Azure DevOps to create and monitor defects or bugs, issues, or other work items. After you configure Azure DevOps on Rational® Test Automation Server, you can create work items for the test runs from

Rational® Test Automation Server and view the created work items on Azure DevOps by clicking the links within Rational® Test Automation Server.

See Configuration of Azure DevOps as a change management system.

- **Support for HCL® Compass as a change management system**

You can now create HCL® Compass records for the tests that you run on Rational® Test Automation Server, after you configure the HCL® Compass server as a change management system on Rational® Test Automation Server.

See Configuration of HCL Compass as a change management system.


- **Copying of the generated keys individually**

When you configured Atlassian Jira as a change management system on Rational® Test Automation Server, you were able to copy the public and consumer keys in a single action. However, it was difficult to differentiate the keys when you pasted them in the individual fields on the Jira server configuration page. You can now copy the public and consumer keys separately and use them to configure the Jira server.

- **Support to create defects on tests run by other project members**

Previously, you were able to create defects only on tests that you ran on Rational® Test Automation Server. You can now create defects in the change management system configured in your project on tests that are run by other project members too.

- **Enhanced publishing of association changes in the system model**

Previously, you were able to publish the changes made to the component associations with test resources in the system model from the **Author > Changes** page. You can now review and publish the association or dissociation changes of test resources with components in a system model from the **System Model** page. You can use the  icon on the **System Model** page to publish the association changes.

- **Viewing results data in Grafana**

By using the Grafana plug-in for Rational® Test Automation Server, you can retrieve and explore test log data for tests run from Rational® Test Automation Server on the Grafana dashboard. See [Test results data in Grafana on page 465](#).

Deprecated capabilities

You can find information about the capabilities that are deprecated in this release and are planned for a removal in a future release.



Note: You can contact the IBM Support team if you have any concerns.

- No capabilities were deprecated in this release.

Removed capabilities

No capabilities were removed in this release.

Installing and upgrading

You can find information about the installation and upgrade instructions for Rational® Test Automation Server.

Installing Rational® Test Automation Server

For instructions about installing the software, see [Installation of the server software on page 29](#).

Upgrading of Rational® Test Automation Server

For instructions about upgrading the software, see [Server software upgrade methods on page 71](#).

Known issues

You can find information about the known issues identified in this release of Rational® Test Automation Server.

Table 1. Release documents - Fix list and known issues

Product	Download document	Knowledge Base
Rational® Test Automation Server	Release document	Knowledge articles

Known problems are documented in the download document for each product and in the form of individual technotes in the Support Knowledge Base:

The knowledge base is continually updated as problems are discovered and resolved. By searching the knowledge base, you can quickly find workarounds or solutions to problems.

Contacting IBM Rational Software Support

You can find information about Contacting IBM Rational Software Support.

- For contact information and guidelines or reference materials that you might need when you require support, read the [IBM Support Guide](#).
- For personalized support that includes notifications of significant upgrades, subscribe to [Product notification](#).
- Before you contact IBM Rational Software Support, you must gather the background information that you might need to describe your problem. When you describe a problem to an IBM software support specialist, be as specific as possible and include all relevant background information so that the specialist can help you solve the problem efficiently. To save time, know the answers to these questions:

- What software versions were you running when the problem occurred?
- Do you have logs, traces, or messages that are related to the problem?
- Can you reproduce the problem? If so, what steps do you take to reproduce it?
- Is there a workaround for the problem? If so, be prepared to describe the workaround.

Chapter 2. System Requirements

This document includes information about hardware and software requirements for IBM® Rational® Test Automation Server.

Contents

- [Hardware on page 13](#)
- [Operating systems and Containers on page 13](#)
 - [Operating systems on page 14](#)
 - [Containers on page 14](#)
- [Host prerequisites on page 15](#)
 - [Licensing on page 15](#)
 - [Web browsers on page 15](#)
- [Supported software on page 16](#)
 - [Development tools on page 16](#)
 - [Integrations on page 17](#)
 - [Runtime environment on page 17](#)
- [Disclaimers on page 12](#)

Disclaimers

This report is subject to the Terms of Use (<https://www.ibm.com/legal/us/en/>) and the following disclaimers:

The information contained in this report is provided for informational purposes only. While efforts were made to verify the completeness and accuracy of the information contained in this publication, it is provided AS IS without warranty of any kind, express or implied, including but not limited to the implied warranties of merchantability, non-infringement, and fitness for a particular purpose. In addition, this information is based on IBM's current product plans and strategy, which are subject to change by IBM without notice. IBM shall not be responsible for any direct, indirect, incidental, consequential, special or other damages arising out of the use of, or otherwise related to, this report or any other materials. Nothing contained in this publication is intended to, nor shall have the effect of, creating any warranties or representations from IBM or its suppliers or licensors, or altering the terms and conditions of the applicable license agreement governing the use of IBM software.

References in this report to IBM products, programs, or services do not imply that they will be available in all countries in which IBM operates. Product release dates and/or capabilities referenced in this presentation may change at any time at IBM's sole discretion based on market opportunities or other factors, and are not intended to be a commitment to future product or feature availability in any way. The underlying database used to support these reports is refreshed on a weekly basis. Discrepancies found between reports generated using this web tool and other IBM documentation sources may or may not be attributed to different publish and refresh cycles for this tool and other sources. Nothing contained in this report is intended to, nor shall have the effect of, stating or implying that any activities undertaken by you will result in any specific sales, revenue growth, savings or other results. You assume sole responsibility for any results you obtain or decisions you make as a result of this report.

Notwithstanding the Terms of Use (<https://www.ibm.com/legal/us/en/>), users of this site are permitted to copy and save the reports generated from this tool for such users own internal business purpose. No other use shall be permitted.

Hardware

You can find information about the hardware requirements for IBM® Rational® Test Automation Server.

Hardware	Requirement	Notes
Disk space	256 GB	Additional resources are required by the cluster for the execution of test assets. Disk space requirements can grow in line with quantity of test assets and results stored.
Memory	32 GB	Additional resources are required by the cluster for the execution of test assets.
Processor	8 CPUs	Additional resources are required by the cluster for the execution of test assets.
Network	Gigabit (1000) Ethernet	
Other hardware		Operating system support is specified by the supported platforms: Kubernetes and Red Hat OpenShift. Refer to their associated documentation for a list of supported operating systems.

Related information

[System Requirements on page 12](#)

Operating systems and Containers

You can find details about the supported operating systems and containers.

Contents

- [Operating systems on page 14](#)
- [Containers on page 14](#)

Bit version support

Different parts of a product might run on the same operating system but support different application bitness.

For example, one part of the product might run only in 32-bit mode, whereas another might support 64-bit tolerate mode.

Bitness	Description
64-Tolerate	The product or part of the product runs as a 32-bit application in the 64-bit platforms listed as supported.
64-Exploit	The product or part of the product runs as a 64-bit application in the 64-bit platforms listed as supported.

Operating systems

Operating system	Hardware	Bitness	IBM® Rational® Test Automation Server	Notes
Ubuntu Server 18.04 LTS	x86-64	64-Exploit	✓	Running K3s v1.25.0
Ubuntu Server 20.04 LTS	x86-64	64-Exploit	✓	Running K3s v1.25.0

Containers

You can find details about the supported containers.

Container	Runtime	Version	Notes
IBM® Rational® Test Automation Server	Red Hat OpenShift Container Platform	4.10	The Container Platform that is available on IBM Cloud is 4.10.32.
		4.11	The Container Platform that is available on IBM Cloud is 4.11.4.

Cloud platforms

You can find details about the supported cloud platforms.

Product	Cloud platform	Version
IBM® Rational® Test Automation Server	Azure Kubernetes Service	1.23.8
		1.24.6

Related information

[System Requirements on page 12](#)

Host prerequisites

You can find the prerequisites that support the operating capabilities for IBM® Rational® Test Automation Server.

Contents

- [Licensing on page 15](#)
- [Web browsers on page 15](#)

Licensing

License Server	Version	IBM® Rational® Test Automation Server	Notes
IBM Common License Key Server	9	✓	

Web browsers

Browser	Version	IBM® Rational® Test Automation Server
Apple Safari	13 or later	✓
Google Chrome	105 or later	✓

Browser	Version	IBM® Rational® Test Automation Server
Microsoft Edge	105 or later	✓
Mozilla Firefox	105 or later	✓
Mozilla Firefox ESR	91 or later	✓

Related information

[System Requirements on page 12](#)

Supported software

You can find details about the additional software that are supported.

Contents

- [Development tools on page 16](#)
- [Integrations on page 17](#)
- [Runtime environment on page 17](#)

Development tools

Supported software	Version	IBM® Rational® Test Automation Server
Git	2.19.0	✓
	2.21.0	✓
	2.22.0	✓
	3.1.0	✓
	3.3.15	✓

Integrations

Supported software	Version	IBM® Rational® Test Automation Server
Azure Plug-ins	Cloud version, latest	✓
HCL® Compass	2.2	✓
IBM® Engineering Test Management	7.0.1	✓
	7.0.2	✓
Jenkins	2.x	✓
Jira	8.x	✓
UrbanCode™ Deploy	7.1	✓
	7.2.1	✓
	7.2.3	✓
Xray app for Jira	Cloud version, latest	✓

Runtime environment

Supported software	Version	IBM® Rational® Test Automation Server	Notes
Helm CLI	3.9.4	✓	Required locally for Helm.
	3.10.1	✓	
	3.10.2	✓	

Supported software	Version	IBM® Rational® Test Automation Server	Notes
Jaeger Operator		✓	Can be optionally installed if you require trace information about test runs. The version depends on the target platform.
Red Hat OpenShift Container Platform CLI (oc)	4.10	✓	Required locally for Red Hat OpenShift Container Platform.
	4.11	✓	
Red Hat OpenShift Container Platform (OCP)	4.10	✓	The Container Platform that is supported by Red Hat OpenShift 4.10.32 on IBM Cloud.
	4.11	✓	The Container Platform that is supported by Red Hat OpenShift 4.11.4 on IBM Cloud.
Red Hat OpenShift Container Storage (OCS)	4.10	✓	The storage class must support ReadWriteMany (RWX) so that executions can be performed on all nodes.
	4.11	✓	

Related information

[System Requirements on page 12](#)

Chapter 3. Getting Started Guide

This guide provides an overview of IBM® Rational® Test Automation Server. You can find the task flows to get you started with Rational® Test Automation Server. This guide is intended for new users.

Before you can perform the various tasks described in the *Getting Started Guide* and the other guides, you must install Rational® Test Automation Server. See [Installation of the server software on page 29](#).

IBM® Rational® Test Automation Server overview

IBM® Rational® Test Automation Server brings together test data, test environments, and test runs and reports into a single, web-based browser for testers and non-testers.

IBM® Rational® Test Automation Server provides the following capabilities:

Web-based continuous testing platform

Rational® Test Automation Server is a web-based continuous testing platform built on modern, cloud native technologies that enables test teams to run a breadth of tests that includes API, functional, and performance tests and to benefit from a holistic view of test progress. Rational® Test Automation Server is built on Docker for easy deployment. You can scale your testing on the cloud or remote on premise systems with native Docker, Kubernetes, and IBM® Red Hat OpenShift support.

Role-based access and security

Security is a key concern for IBM clients and therefore, Rational® Test Automation Server brings a comprehensive, role-based access control scheme to the server with project owners assigning key permissions (by using roles) for specific members, for example managing test data or working with secrets such as passwords.

Running of tests from the server by using Docker containers

Server-based running of tests is the starting point for Rational® Test Automation Server. For members of a project with the appropriate role, Rational® Test Automation Server enables direct running of tests from the browser by using transient Docker containers.

Connected agents for existing performance agents

Agent owners can connect existing performance agents to the server and add them to a project for running schedules and Accelerated Functional Testing (AFT) Suites on the current infrastructure.

Project overview statistics

The **Overview** page for Rational® Test Automation Server offers you a quick, simple view on the state of testing for your projects.

Project home page

The home page lists your projects and other projects, which makes it easy to manage different projects within an organization.

Reporting and the Resource Monitoring Service

Rational® Test Automation Server provides the home for capabilities that previously were hosted on Rational® Test Control Panel. Reporting and the Resource Monitoring Service are in Rational® Test Automation Server and provide a more direct relationship with their related projects. These capabilities also benefit from the project level, role-based access controls. You can view unified test results to help you make informed business decisions.

Test data authoring

Beyond the concept of a project held in a Git repository for a simple location of tests and related assets, you can do full concurrent editing of test data sets directly from Rational® Test Automation Server.

This true multiuser capability enables team members to collaborate more easily as well as try out data changes without impacting the rest of the team. When satisfied with the results, team members can push their changes.

Integration with DevOps tools

You can integrate Rational® Test Automation Server with various popular DevOps tools like Jenkins, UrbanCode™ Deploy or Microsoft Azure DevOps to get more value from your DevOps pipelines. You can use UrbanCode™ Deploy to define a deployment process that automatically triggers test cases and have those test insights available directly within IBM® UrbanCode™ Velocity. With the Microsoft Azure DevOps integration, you can also define a Azure DevOps pipeline that includes direct execution of tests by using the Rational® Test Automation Server scalable infrastructure. Rational® Test Automation Server also integrates with Jira for defect tracking and GitHub for software source management and version control.

Supported versions of assets and resources

You can find information about the versions of assets, resources, agents and Dockers that are supported in Rational® Test Automation Server. You can also find information about the versions of the desktop clients that are supported.

The following table lists the versions of assets or resources created in the desktop clients along with the agents, Dockers, or proxies that are supported in Rational® Test Automation Server:

Resource	Supported	Not supported
Assets or resources created in IBM® Rational® Functional Tester	Assets or resources created in Rational® Functional Tester which are of versions earlier or the same as Rational® Test Automation Server.	Assets or resources created in any later version of Rational® Functional Tester that is later than the version of Rational® Test Automation Server.
Assets or resources created in IBM® Rational® Integration Tester	Assets or resources created in Rational® Integration Tester which are of versions earlier or the same as Rational® Test Automation Server	Assets or resources created in any later version of Rational® Integration Tester that is later than the version of Rational® Test Automation Server.

Resource	Supported	Not supported
Assets or resources created in IBM® Rational® Performance Tester	Assets or resources created in Rational® Performance Tester which are of versions earlier or the same as Rational® Test Automation Server	Assets or resources created in any later version of Rational® Performance Tester that is later than the version of Rational® Test Automation Server.
IBM® Rational® Performance Tester Agents	Rational® Performance Tester Agents which is the same version as Rational® Test Automation Server	Before you install the agents that are of the current version, see Management of performance agents on page 210 .
Dockers on a remote host computer	Image of Rational® Test Automation Server which is the same version as Rational® Test Automation Server that you want to use to run tests, must be installed on the Docker on a remote host computer.	Before you set up Dockers remotely, see Management of Docker hosts on page 218 .
HTTP proxies	Installation of the proxy component from Rational® Test Control Panel which is the same version as Rational® Test Automation Server.	Before you use HTTP proxies, see Setting up the HTTP proxy on page 413 .

If there is a mismatch in the versions of the assets, resources, agents, or images used in Dockers with the version of Rational® Test Automation Server that you are using, any of the following events can occur:

- Warnings or Errors are displayed when you add a repository to a project in your team space.
- Warnings or Errors are displayed when you open a project that you migrated from a previous version of Rational® Test Automation Server.
- Warnings or Errors are displayed when you add or manually refresh a repository to which you added assets or resources that were created on a desktop client with a version previous to or later than the version of Rational® Test Automation Server that you want to use.
- Test runs that you start or schedule on an agent or Docker fail to run.
- You cannot view the proxies that are registered with Rational® Test Automation Server on the **Agents and Intercepts** page.

Suites and tests that are supported on Rational® Test Automation Server

You can find information about the types of Test Suites and tests that you can configure for a run in a project on Rational® Test Automation Server.

The following table lists the Test Suites and test types that are supported and those tests that are not supported to run on Rational® Test Automation Server:

Supported Suites	Supported tests	Unsupported tests
<ul style="list-style-type: none"> • Accelerated Functional Testing (AFT) Suites • Compound Tests that contain any of the following types of tests: <ul style="list-style-type: none"> ◦ Web UI tests ◦ HTML tests ◦ Mobile tests 	<ul style="list-style-type: none"> • Functional Web UI tests • Functional Windows tests • Functional Mobile tests 	<ul style="list-style-type: none"> • Traditional HTML tests • Traditional non-HTML tests
<ul style="list-style-type: none"> • API Suites that include the following Test Suites: <ul style="list-style-type: none"> ◦ Test Suites that contain scenarios with references to local stubs. ◦ Test Suites that contain stubs that virtualize services run in any Kubernetes cluster or in the Kubernetes cluster that hosts Rational® Test Automation Server. • Stubs 	API tests	Suites or tests with subscribe actions that operate in the watch mode.
<ul style="list-style-type: none"> • Compound Tests that contain performance tests • Rate Schedules • VU Schedules 	Performance tests	<ul style="list-style-type: none"> • A single test script of any test extension. • Tests that belong to 32-bit test extensions and SOA Quality included in VU Schedule, Rate Schedule, and Compound Test. • Citrix, SAP, or Siebel as an individual test or as part of a Compound Test.
Postman resources		

Supported Suites	Supported tests	Unsupported tests
JMeter tests		
JUnit tests		



Note: You must create compound tests for the same type of tests. For example, you can create a compound test that has different mobile tests that use different mobile testing platforms.



Restriction: You cannot merge folders that contain different test projects in the same project directory. For example, you cannot create a project directory that contains a Performance test project and an API Suite project. You must create separate project directories for each test type. For example, you must create the Performance test in the project directory as `/MyPerfProject/` and the API Suite in the project directory as `/MyAPIProject`, before you commit the projects to the remote repository.

Related information

[Test run configurations on page 249](#)

Virtual services that are supported on Rational® Test Automation Server

You can find information about the virtual services that are supported on Rational® Test Automation Server.

You can start a running instance of the virtual services from Rational® Test Automation Server to run on the following locations:

- The default Kubernetes cluster of Rational® Test Automation Server.
- A remote Kubernetes cluster.

The default Kubernetes cluster or a remote Kubernetes cluster supports running of the following types of virtual services:

- Virtual services that utilize the WebSphere® MQ transport.
- Virtual services that utilize the HTTP transport.
- Virtual services that virtualize the Istio based services. You can run virtual services for the following types of requests received or sent by the Istio service mesh:
 - Requests received by services in the Istio service mesh.
 - Requests sent from namespaces in the Istio service mesh to external services that are not in the Istio service mesh.



Restriction: You cannot run virtual services on a remote Docker host.

Related information

[Management of virtualized services on page 408](#)

Task flow: Test runs and results in IBM® Rational® Test Automation Server

You can use the task flow diagram to get started with IBM® Rational® Test Automation Server. After you install the software, you can complete the tasks in sequence to run test assets in Rational® Test Automation Server and view and analyze the test results.

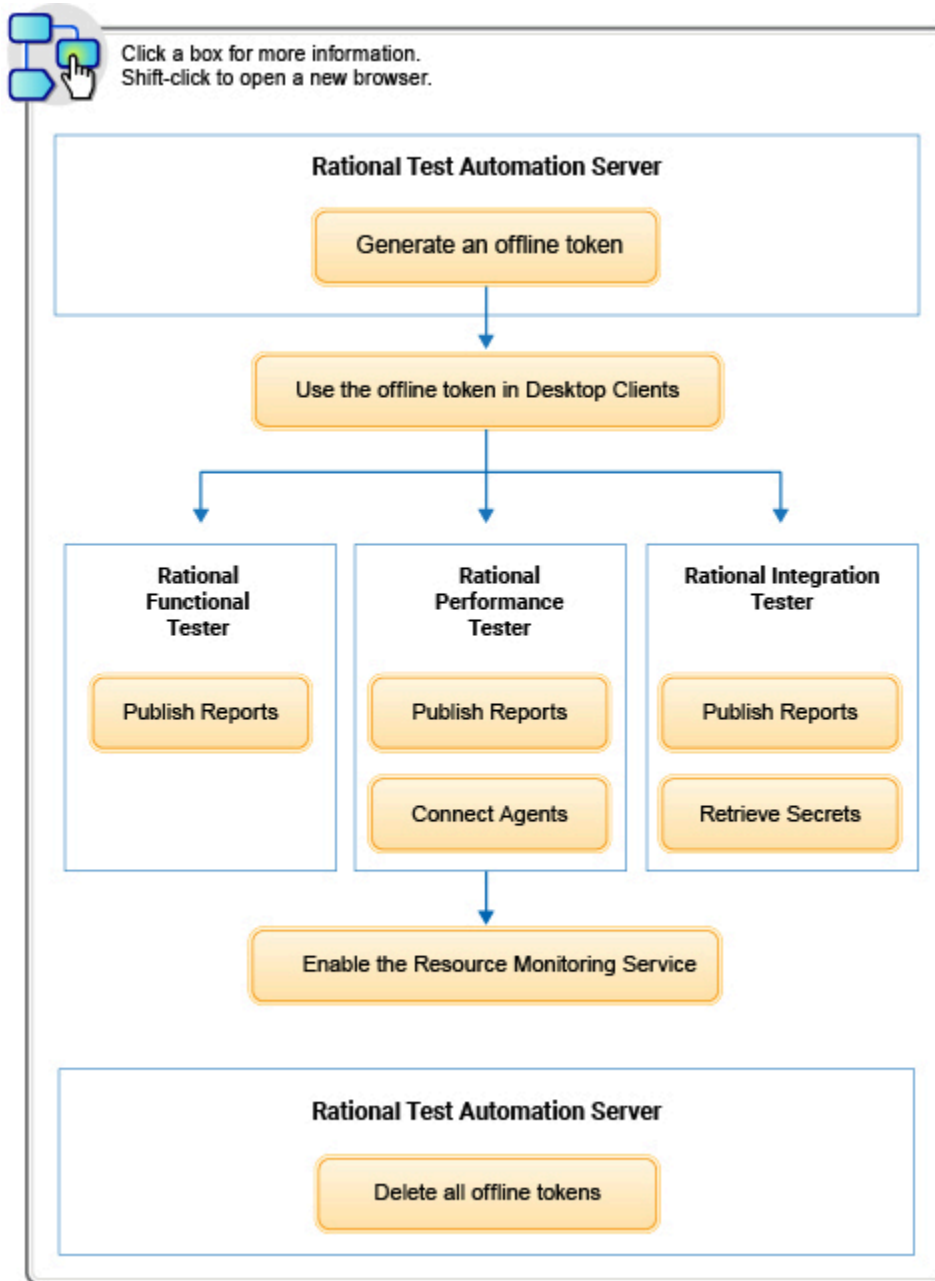
You can click the links to get more information about the tasks. The diagram also provides the tasks to be performed by personas. Rational® Test Automation Server supports user roles.



1. Default user administration on page 107
2. Test assets and a server project on page 616
3. Adding a project on page 619
4. Adding repositories to a server project on page 620
5. Protecting API test assets by using secrets on page 640
6. Managing an encrypted dataset on page 646
7. Adding users to a project on page 623
8. Adding a performance agent to a project on page 213
9. Test run configurations on page 249

Task flow: IBM® Rational® Test Automation Server access from desktop clients

You can use the task flow diagram to help you access the server from the different desktop clients to retrieve secrets, publish reports, or enable resource monitoring agents. You can perform these tasks after you install and set up the server.



1. [Managing access to IBM Rational Test Automation Server on page 600](#)
2. [Test results and reports on page 456](#)
3. [Test results and reports on page 456](#)
4. [Test results and reports on page 456](#)
5. [Retrieving secrets](#)
6. [Managing access to IBM Rational Test Automation Server on page 600](#)
7. [Managing access to IBM Rational Test Automation Server on page 600](#)
8. [Installing Rational Performance Tester Agent](#)
9. [Enabling of Resource Monitoring services for a schedule](#)

Related information

[Managing access to server projects on page 627](#)

Accessibility features

Accessibility features help users who have physical disabilities, such as visual and, hearing impairment, or limited mobility, to use the software products successfully.

Accessibility compliance

To understand the accessibility compliance status for IBM® Rational® Test Automation Server, refer to [Accessibility Conformance Reports](#).

For more information about IBM and accessibility, refer to [IBM Accessibility](#).

The product documentation is published by using Oxygen XML WebHelp Responsive. To understand the accessibility compliance status for Oxygen XML WebHelp Responsive, refer to [WebHelp Responsive VPAT Accessibility Conformance Report](#).

Accessing UI elements

Rational® Test Automation Server supports navigation in the UI by using different methods such as a mouse, keyboard, or touchpad.

You can use the keyboard keys such as **Tab**, arrow keys such as **UP**, **DOWN**, **LEFT**, and **RIGHT** to navigate to the different pages in the **Navigation** pane or to the different action labels in the right pane on the UI.

Chapter 4. Administrator Guide

This guide describes how to install the IBM® Rational® Test Automation Server software.

After you install the software, you can perform administration tasks such as license configuration, user management, security, memory and disk usage management, back up and restore user data, and other tasks that a server administrator can perform. This guide is intended for administrators.

As a *Server Administrator*, you can assign the role of a *Server Administrator* to any user. For information about how to change the role, see [Default user administration on page 107](#).



Remember: When your role is changed from a user to a *Server Administrator*, you must log out and log in again so that Rational® Test Automation Server can apply the change in your role.

Installation of the server software

To get started working with IBM® Rational® Test Automation Server, you must first install the server software.

You can install Rational® Test Automation Server on the following platforms:

- IBM® Red Hat OpenShift
- Ubuntu Server (by using k3s)
- Azure Kubernetes Service (AKS)

Installation task flows

IBM® Red Hat OpenShift Container platform

You must perform the following tasks to install Rational® Test Automation Server:

1. Set up a IBM® Red Hat OpenShift Container Platform (OCP).
2. Install the Jaeger operator to trace test logs and Jaeger-based reports when you run tests.
3. Install the OCP command-line interface (CLI) and the Helm software.
4. Install Rational® Test Automation Server.

Ubuntu Server

You must perform the following tasks to install Rational® Test Automation Server:

1. Set up Ubuntu Server.
2. Install the OpenSSH server and the Helm software.

3. Set up the Kubernetes environment (k3s) by using the provided script.
4. Install Rational® Test Automation Server.

Azure Kubernetes Service

You must perform the following tasks to install Rational® Test Automation Server:

1. Create an Azure subscription and set up the AKS cluster.
2. Install the Helm software.
3. Install the kubectl and Azure command-line interface (Azure CLI) software.
4. Install Emissary-ingress.
5. Install Rational® Test Automation Server.

To learn more about the installation of the server software on platforms such as IBM® Red Hat OpenShift, AKS, and Ubuntu, refer to the following topics:

Installation of the server software on IBM® Red Hat OpenShift

You can find information about the tasks that you can perform to install the Rational® Test Automation Server software on the IBM® Red Hat OpenShift platform.

If you have a Red Hat OpenShift Container Platform (OCP) cluster with no access to the internet, you can install Rational® Test Automation Server by using an air-gapped environment.

Prerequisites for installing the server software on IBM® Red Hat OpenShift

You must complete certain tasks before you install Rational® Test Automation Server on the Red Hat OpenShift platform.

The following sections describe each prerequisite in detail:

- [Entitlement key on page 30](#)
- [Red Hat OpenShift platform on page 31](#)
- [Mandatory software on page 31](#)
- [Persistent storage on page 31](#)
- [Service virtualization usage metrics on page 32](#)

Entitlement key

You must copy the **Entitlement key** from the [Container software library](#) to pull the images that are used by the server software. For more information, refer to [Creating an API key](#).

Red Hat OpenShift platform

You must set up the Red Hat OpenShift platform, and then install the following components in your IT infrastructure:

- Red Hat OpenShift Container Platform (OCP). For more information, refer to the Installing section in the [Red Hat OpenShift](#) documentation.
- Dynamic Volume Provisioning that supports ReadWriteOnce (RWO) and ReadWriteMany (RWX) access modes. For more information, refer to the Dynamic provisioning section in the [Red Hat OpenShift](#) documentation.
- **Optional:** Jaeger operator to trace test logs and Jaeger-based reports when you run tests. For more information, refer to [Installing the Operators](#).
- **Optional:** You can configure the Red Hat OpenShift software-defined networking (SDN) to provide a unified cluster network that enables the communication between pods across the Red Hat OpenShift Container Platform cluster. The default installation process includes the NetworkPolicy resources and these resources are acted upon only if you configure the SDN appropriately. For more information, refer to the Red Hat OpenShift SDN default CNI network provider section in the [Red Hat OpenShift](#) documentation.

You must have access to the Red Hat OpenShift cluster with cluster administrator privileges. One of the ways to get access to the Red Hat OpenShift cluster is to run the following command on the Red Hat OpenShift command-line interface:

```
oc login -u kubeadmin -p {password} https://api.{openshift-cluster-dns-name}:6443
```



Note: For more information about specific versions of software requirements, see [System Requirements on page 14](#).

Mandatory software

You must install the following software on Red Hat OpenShift:

- Helm. For more information, refer to the Installing a Helm chart on Red Hat OpenShift Container Platform cluster section in the [Red Hat OpenShift](#) documentation.
- Red Hat OpenShift CLI. For more information, refer to the Getting started with the CLI section in the [Red Hat OpenShift](#) documentation.

Persistent storage

Rational® Test Automation Server requires persistent storage in addition to the minimum resources required by the cluster. Depending on the workload, you might require additional persistent storage.

The default configuration creates the PersistentVolumeClaim that dynamically provision the PersistentVolume. The following table lists the PersistentVolumeClaim, its size, and access mode during the configuration:

PersistentVolumeClaim	Size	Access Mode	Content
data-{my-rtas}-postgresql-0	26 GB	ReadWriteOnce CLI Abbreviation: RWO	User data used by services
data-{my-rtas}-rabbitmq-0	8 GB	RWO	Notifications between services
data-{my-rtas}-results-0	8 GB	RWO	Reports
data-{my-rtas}-tam-0	32 GB	RWO	Cloned git repositories
data-{my-rtas}-userlibs-0	1 GB	ReadWriteMany CLI Abbreviation: RWX	User-provided third-party libraries and extensions



Note: {my-rtas} is the name of the release that you provide during the installation of the server software.

Service virtualization usage metrics

You must perform the following tasks so that Rational® Test Automation Server shows the correct usage of service virtualization metrics:

- Create the cluster-monitoring-config ConfigMap object to configure the Red Hat OpenShift Container Platform (OCP) monitoring stack, if the ConfigMap does not exist.

For more information about creating a cluster monitoring config map, based on the version of OCP that you use, refer to [Creating a cluster monitoring config map](#).

- Enable the user workload monitoring services in the Red Hat OpenShift cluster to show the service virtualization usage metrics in Rational® Test Automation Server.

For more information about the enabling monitoring of your services, based on the version of OCP that you use, refer to [Enabling monitoring of your services](#)

Related information

[Red Hat OpenShift Container Platform 4.7 Documentation](#)

[Red Hat OpenShift Container Platform 4.8 Documentation](#)

[System Requirements on page 12](#)

Installing the server software on IBM® Red Hat OpenShift


You can install Rational® Test Automation Server on the Red Hat OpenShift platform that has the Kubernetes environment to run functional, integration, and performance tests. Rational® Test Automation Server combines test data, test environments, and test runs and reports into a single web-based browser for testers and non-testers.




Before you begin

- You must have read and been familiar with additional Helm parameters. See [Additional Helm parameters on page 69](#).
- You must have performed the following tasks:
 - Completed tasks provided in the Prerequisites section. See [Prerequisites for installing the server software on IBM Red Hat OpenShift on page 30](#).
 - Installed the following software:
 - Red Hat OpenShift CLI
 - Helm
 - Copied the **Entitlement key** from the [Container software library](#).

About this task

The following table lists the variables that you must replace with the actual value in the command.

Variables	Description
{your_entitlement_key}	The key that you copied from the Container software library .
{example@abc.com}	The email address of the administrator if required.
{my-rtas}	<p>The release name of your choice.</p> <p> Note: The release name must consist of alphanumeric characters that are in lowercase or "-" (hyphen). The release name must also start with an alphabetic character and end with an alphanumeric character. For example, <i>my-org</i> or <i>abc-123</i>.</p>
{openshift-cluster-dns-name}	The ingress DNS name that you selected for the server.

Variables	Description
	<p> Remember: You must provide the value that consists of alphanumeric characters that are in lowercase, "-" (hyphen) or "." (period). The value must also start and end with an alphanumeric character.</p> <p>You can run the following command to obtain the default value of {openshift-cluster-dns-name}:</p> <pre>oc get --namespace=openshift-ingress-operator ingresscontroller/default -ojsonpath='{.status.domain}'</pre>
{password-seed}	<p>A value of your choice for the password.</p> <p> Important: This password seed is used to create several default passwords for the server. You must store the password seed securely. When you install the server software by using the backup of the user data, you can reuse the password seed. You can use this seed to restore the backed-up files either on the current or later versions of the server software.</p>
{rlks-ip-address}	<p>The IP address of License Key Server, if you want to set the license value for the first time.</p> <p> Important: When you upgrade the product from the previous version, you must configure the value of License Key Server in the Team Space License Configuration window when the installation of the server is complete.</p>
{my-jaeger-dashboard-url}	<p>The URL of the Jaeger server.</p>

1. Log in to your OCP cluster as a cluster administrator by running the oc login command.
2. Create a namespace in which you want to install the server software by running the following command:

```
oc new-project test-system
```



Remember: The `test-system` is the name of the namespace. If you created a namespace by using a different value, then you must use that value in place of `test-system` in all the instances in this procedure.

3. Add the entitlement registry to Helm to access the server install charts by running the following command:

```
helm repo add ibm-helm https://raw.githubusercontent.com/IBM/charts/master/repo/ibm-helm
```

4. Run the following command to get the latest updates from the repository:

```
helm repo update
```

5. Run the following command to retrieve the charts:

```
helm pull --untar ibm-helm/ibm-rtas-prod --version 10.1051.0
```

6. Create a Secret to pull images that are used by Rational® Test Automation Server by running the following commands:

```
oc create secret docker-registry cp.icr.io \
-n test-system \
--docker-server=cp.icr.io \
--docker-username=cp \
--docker-password={your_entitlement_key} \
--docker-email={example@abc.com}
```

7. Perform the following steps to add the Certificate Authority (CA) into a Secret and enable certificates as trusted certificates:

- a. Run the following command to verify whether an additional CA is required:

```
curl -sw'{http_code}' -o/dev/null \
"https://wildcard.${oc get -n openshift-ingress-operator ingresscontroller default
-ojsonpath='{.status.domain}')"
```

If the result of the command is displayed as `503`, the CA is already trusted. You must continue with [step 8 on page 36](#).

If the result of the command is displayed as `000`, then CA must be added into a Secret. You must continue with [step 7.b on page 35](#).

- b. Run the following command to get the default CA in a PEM format:

```
oc get -n openshift-ingress-operator secret router-ca -ojsonpath='{.data.tls.crt}' |
base64 --decode > ca.crt
```

- c. Run the following command to validate that the CA used to sign the certificate is the same for ingress:

```
curl -sw'{http_code}' -o/dev/null --cacert ca.crt \
"https://wildcard.${oc get -n openshift-ingress-operator ingresscontroller default
-ojsonpath='{.status.domain}')"
```

If the result of the command is displayed as `503`, then you must continue with next step.

If the result of the command is displayed as `000`, then the configuration of the certificate has been customized. You must find the signer of the certificate to continue with the next step.

- d. Run the following command to create an ingress Secret to store the CA:

```
oc create secret generic -n test-system ingress --from-file=ca.crt=ca.crt
```

When some components such as static agents or Docker agents want to communicate with Rational® Test Automation Server, the component presents its certificate to the server to verify its identity. Rational® Test Automation Server trusts the component only if it is signed by a recognized and trusted CA. Therefore, you must add the signed CA into a trust by placing it in a Secret to enable certificates as trusted certificates.

8. Perform the following steps to install the server software:

- a. Run the following commands to update the runAsUser and fsGroup to match the Security Context Constraints (SCCs):

```
sed -i -e "s/runAsUser: 1001/runAsUser: $(oc get project test-system -oyaml \
| sed -r -n 's# *openshift.io/sa.scc.uid-range: *([0-9]*)/.*\#1#p')/g;
s/fsGroup: 1001/fsGroup: $(oc get project test-system -oyaml \
| sed -r -n 's# *openshift.io/sa.scc.supplemental-groups: *([0-9]*)/.*\#1#p')/g"
ibm-rtas-prod/values-openshift.yaml
```

Rational® Test Automation Server is compatible with the restricted SCC. You must run this command to ensure that the runAsUser and fsGroup strategies match with the SCC policy.

- b. Perform one of the steps described in the following table to install the server software based on your requirement:

Step description	Step no
To install the server software	Perform 8.b.i on page 37
To install the server software and enable Jaeger for performance and Web UI tests logs	Perform 8.b.ii on page 37



Important: When you install the server software on Red Hat OpenShift 4.11, following warning can be ignored:

```
W1027 17:09:25.719131 17132 warnings.go:70] would violate PodSecurity "restricted:v1.24":
seccompProfile (pod or container "main" must set securityContext.seccompProfile.type to
"RuntimeDefault" or "Localhost")
```

The warning is displayed because the OpenShift *restricted-v2* security context constraints (SCC) policy requires that the **seccompProfile** parameter must be set to *RuntimeDefault* or is to be left empty. When the value is not set, the value defaults to *RuntimeDefault*. The empty parameter is compatible with the *restricted-v1* SCC policy and works on previous versions of Red Hat OpenShift 4.11.

To prevent this warning, you can explicitly set the value of the **seccompProfile** parameter to *RuntimeDefault*.



Remember:



- If you upgrade the product from the previous version, then you must use the same value for `global.persistence.rwxStorageClass` parameter that you used in the previous installation.

You can run the following command to obtain the value that you used for `global.persistence.rwxStorageClass`:

```
oc get pvc -n test-system data-{my-rtas}-userlibs-0 \
  -ojsonpath='{.spec.storageClassName}' && echo
```

- The default certificate that terminates TLS connections has a single wildcard. Therefore, you must prefix a single hostname segment for the `global.ibmRtasIngressDomain` parameter.

For example, `--set global.ibmRtasIngressDomain=rtas.{dns name of the openshift cluster}`

- To provide enhanced security to Rational® Test Automation Server, the **Sign up** link on the **Login** page of Rational® Test Automation Server is not visible when you install the server software.

If you want to enable the **Sign up** link on the **Login** page, then you must add the `--set keycloak.signup.enabled=true` parameter in any of the following helm install commands.

- i. Run the following command to install the server software:

```
helm install {my-rtas} ./ibm-rtas-prod -n test-system \
--set license=true \
-f ibm-rtas-prod/values-openshift.yaml \
--set global.persistence.rwxStorageClass=ibmc-file-gold \
--set global.ibmRtasIngressDomain=rtas.{openshift-cluster-dns-name} \
--set global.ibmRtasPasswordAutoGenSeed={password-seed} \
--set global.ibmRtasRegistryPullSecret=cp.icr.io \
--set global.rationalLicenseKeyServer=@{rlks-ip-address}
```

- ii. Run the following command to install the server software and to enable Jaeger for performance and Web UI tests logs:

```
helm install {my-rtas} ./ibm-rtas-prod -n test-system \
--set license=true \
-f ibm-rtas-prod/values-openshift.yaml \
--set global.persistence.rwxStorageClass=ibmc-file-gold \
--set global.ibmRtasIngressDomain=rtas.{openshift-cluster-dns-name} \
--set global.ibmRtasPasswordAutoGenSeed={password-seed} \
--set global.ibmRtasRegistryPullSecret=cp.icr.io \
--set global.rationalLicenseKeyServer=@{rlks-ip-address} \
--set-string execution.annotations.sidecar\\.jaegertracing\\.io/inject=true \
--set global.jaegerAgent.internalHostName=localhost \
--set global.jaegerDashboard.externalURL={my-jaeger-dashboard-url}
```

9. **Optional:** Run the following command to remove a job that is used to initialize the PostgreSQL database during the installation of the server software:

```
oc delete job {my-rtas}-postgresql-init -n test-system
```

10. Run the following script from the `ibm-rtas-prod/files` directory to verify and test the installed server software:

```
ibm-rtas-prod/files/helm-test-diag.sh {my-rtas} -n test-system
```

Results

You have installed the server software. The command line displays the following information:

- Keycloak URL to manage and authenticate users.
- A URL to access the Rational® Test Automation Server UI.

What to do next

You can perform the following tasks:

- Configure licenses for team spaces. See [Configuring licenses for team spaces on page 139](#).
- Configure the server software. See [Configuration of the server software on page 105](#).

You can migrate the user data if you upgraded the server software from 10.1.2 or earlier to a new version. See [Migrating user data on Red Hat OpenShift on page 85](#).

Related information

[Troubleshooting on page 663](#)

Installation of the server software in an air-gapped environment

You can find information about the tasks that you can perform to install Rational® Test Automation Server on the IBM® Red Hat OpenShift platform in an air-gapped environment.

If your Red Hat OpenShift Container Platform (OCP) cluster is within a restricted network with no access to the internet, then you can install Rational® Test Automation Server on such OCP clusters by using an air-gapped environment. To install Rational® Test Automation Server, unlike online installations, air-gapped installations require an internal registry to mimic a typical online installation by using images from your internal registry.

In an air-gapped environment, you can mirror the images of Rational® Test Automation Server from IBM Entitled Registry to the internal Docker registry that has access to the OCP cluster. With the help of a Bastion host that can connect to the internet and your OCP cluster, you can install Rational® Test Automation Server on the OCP cluster.

Prerequisites for installing the server software in an air-gapped environment

You must complete certain tasks before you install Rational® Test Automation Server in an air-gapped environment on the Red Hat OpenShift platform.

The following sections describe each prerequisite in detail:

- [Entitlement key on page 39](#)
- [Red Hat OpenShift Container Platform on page 39](#)
- [Bastion host on page 39](#)
- [Docker registry on page 40](#)
- [Mandatory software on page 40](#)
- [Service virtualization usage metrics on page 40](#)

Entitlement key

You must copy the **Entitlement key** from the [Container software library](#) to pull the images that are used by the server software. For more information, refer to [Creating an API key](#).

Red Hat OpenShift Container Platform

You must install Red Hat OpenShift Container Platform (OCP) and have cluster administrator privileges. For more information, refer to the [Red Hat OpenShift](#) documentation.

For more information about specific versions of software requirements, see [System Requirements on page 12](#).

You can run the following command on the Red Hat OpenShift command-line interface to gain cluster administrator access:

```
oc login -u kubeadmin -p {password} https://api.{openshift-cluster-dns-name}:6443
```

You can run the following command to obtain the default value of `openshift-cluster-dns-name`:

```
oc get --namespace=openshift-ingress-operator ingresscontroller/default -ojsonpath='{.status.domain}'
```

Bastion host

You must set up a Bastion host that has Linux installed. You must also have internet access.

A bastion host is a server that is provisioned with a public IP address and must be accessible through remote access Secure Shell (SSH).



Note: You must ensure that the Bastion host has access to your OCP cluster.

For more information refer to the [IBM Cloud Docs](#).

Docker registry

You must set up a local Docker registry to store the images of Rational® Test Automation Server and ensure that it has access to the OCP cluster and the Bastion host.

For more information refer to the [Red Hat OpenShift](#) documentation.

Mandatory software

You must install the following software on Red Hat OpenShift:

- Red Hat OpenShift command-line interface (CLI) on the Bastion host. For more information, refer to the Getting started with the CLI section in the [Red Hat OpenShift](#) documentation.
- IBM Cloud Pak CLI (cloudctl) on the Bastion host to manage Container Application Software for Enterprises (CASE). For more information, refer to the [cloud-pak-cli](#) repository.
- Helm. For more information, refer to the Installing a Helm chart on an Red Hat OpenShift Container Platform cluster section in the [Red Hat OpenShift](#) documentation.

Service virtualization usage metrics

You must perform the following tasks so that Rational® Test Automation Server shows the correct usage of service virtualization metrics:

- Create the cluster-monitoring-config ConfigMap object to configure the Red Hat OpenShift Container Platform (OCP) monitoring stack, if the ConfigMap does not exist.

For more information about creating a cluster monitoring config map, based on the version of OCP that you use, refer to [Creating a cluster monitoring config map](#).

- Enable the user workload monitoring services in the Red Hat OpenShift cluster to show the service virtualization usage metrics in Rational® Test Automation Server.

For more information about the enabling monitoring of your services, based on the version of OCP that you use, refer to [Enabling monitoring of your services](#)

Related information

[Red Hat OpenShift Container Platform 4.7 Documentation](#)

[Red Hat OpenShift Container Platform 4.8 Documentation](#)

[IBM Cloud Pak foundational services](#)

[System Requirements on page 12](#)

Configuring the Bastion host

You must configure the Bastion host to mirror the images of Rational® Test Automation Server from IBM Entitled Registry to the Red Hat OpenShift Container Platform (OCP) cluster.

Before you begin

You must have performed the following tasks:

- Completed the tasks provided in the Prerequisites section. See [Prerequisites for installing the server software in an air-gapped environment on page 39](#).
- Copied the **Entitlement key** from the [Container software library](#).

1. Log in to your OCP cluster as a cluster administrator by running the `oc login` command.
2. Run the following command to confirm that the container registry is accessible from the Bastion host:

```
oc get routes -n openshift-image-registry
```

3. **Optional:** Create a route to provide external access to the container registry if no resources are found:

```
oc patch configs.imageregistry.operator.openshift.io/cluster \
--patch '{"spec":{"defaultRoute":true}}' --type=merge
```

4. Create a namespace for the internal registry to host the mirror images:

```
oc new-project cp
```



Note: The `cp` is the name of the namespace and it is required to pull the images from IBM Entitled Registry.

5. Create the following environment variables with the image name and download the images of Rational® Test Automation Server to the offline store by running the following commands:

Before mirroring images of Rational® Test Automation Server, you must set the environment variables on your Bastion host, and connect to the internet so that you can download the corresponding CASE files from IBM Entitled Registry.

```
export CASE_NAME=ibm-rtas-case
export CASE_VERSION=10.1051.0
export CASE_ARCHIVE=${CASE_NAME}-${CASE_VERSION}.tgz
export CASE_REMOTE_PATH=https://github.com/IBM/cloud-pak/raw/master/repo/case/${CASE_ARCHIVE}
export OFFLINEDIR=$HOME/offline

cloudctl case save \
--case $CASE_REMOTE_PATH \
--outputdir $OFFLINEDIR
```

6. Run the following commands to store credentials to authenticate IBM Entitled Registry:

The images of Rational® Test Automation Server are within IBM Entitled Registry. You must log in to IBM Entitled Registry to access source images and log in to your internal registry that acts as a local registry when you copy the images.

```
export ENTITLEMENT_KEY=YOUR_ENTITLEMENT_KEY
export CASE_INVENTORY_SETUP=ibmRtasProd

cloudctl case launch \
--case $OFFLINEDIR/$CASE_ARCHIVE \
--inventory $CASE_INVENTORY_SETUP \
--action configure-creds-airgap \
--args "--registry cp.icr.io --user cp --pass $ENTITLEMENT_KEY"
```



Note: You must replace `YOUR_ENTITLEMENT_KEY` with the key that you copied from the [Container software library](#).

7. Run the following commands to store credentials to authenticate the local Docker registry:

```
export LOCAL_DOCKER_REGISTRY=$(oc get route default-route -n openshift-image-registry -o
jsonpath='{.spec.host}')
export LOCAL_DOCKER_USER=UNUSED
export LOCAL_DOCKER_PASSWORD=$(oc whoami -t)

cloudctl case launch \
--case $OFFLINEDIR/$CASE_ARCHIVE \
--inventory $CASE_INVENTORY_SETUP \
--action configure-creds-airgap \
--args "--registry $LOCAL_DOCKER_REGISTRY --user $LOCAL_DOCKER_USER --pass
$LOCAL_DOCKER_PASSWORD"
```

8. Run the following command to mirror the images from IBM Entitled Registry to the local Docker registry:

```
cloudctl case launch \
--case $OFFLINEDIR/$CASE_ARCHIVE \
--inventory $CASE_INVENTORY_SETUP \
--action mirror-images \
--args "--registry $LOCAL_DOCKER_REGISTRY --inputDir $OFFLINEDIR"
```

9. Run the following commands to configure the OCP cluster where Rational® Test Automation Server needs to be installed with a `global pull secret` for the local Docker registry:

By using the `global pull secret`, the OCP cluster can pull the images of Rational® Test Automation Server from your local Docker registry instead of IBM Entitled Registry.

```
cloudctl case launch \
--case $OFFLINEDIR/$CASE_ARCHIVE \
--inventory $CASE_INVENTORY_SETUP \
--action configure-cluster-airgap \
--namespace default \
--args "--registry $LOCAL_DOCKER_REGISTRY --inputDir $OFFLINEDIR"
```



Notes:



- After you perform step 9 on page 42, the cluster restarts all the nodes.
- The `namespace` argument is required for legacy reasons, its value can be any namespace that exists.

10. Run the following command to extract the mirrored images:

```
tar xf $OFFLINEDIR/charts/ibm-rtas-prod-10.1051.0.tgz
```

Results

You have configured the Bastion host and mirrored the images of Rational® Test Automation Server.

What to do next

You can install the server software on your restricted OCP. See [Installing the server software in an air-gapped environment on page 43](#).

Related information

[Using image pull secrets](#)

[Configuring image registry repository mirroring](#)

[IBM Cloud Pak foundational services](#)

[IBM Cloud Pak CLI \(cloudctl\) CASE commands](#)

Installing the server software in an air-gapped environment





You might have an environment that has a high level of security and is isolated from the internet. In such a scenario, you can install Rational® Test Automation Server on the Red Hat OpenShift platform in an air-gapped environment.

Before you begin

- You must have read and been familiar with additional helm parameters. See [Additional Helm parameters on page 69](#).
- You must have performed the following tasks:
 - Completed the tasks provided in the Prerequisites section. See [Prerequisites for installing the server software in an air-gapped environment on page 39](#).
 - Configured the Bastion host. See [Configuring the Bastion host on page 41](#).
 - Copied the **Entitlement key** from the [Container software library](#).

About this task

The following table lists the variables that you must replace with the actual value in the command.

Variables	Description
{openshift-cluster-dns-name}	<p>The ingress DNS name that you selected for the server.</p> <p> Remember: You must provide the value that consists of alphanumeric characters that are in lowercase, "-" (hyphen) or "." (period). The value must also start and end with an alphanumeric character.</p> <p>You can run the following command to obtain the default value of {openshift-cluster-dns-name}:</p> <pre data-bbox="462 630 1425 682">oc get --namespace=openshift-ingress-operator ingresscontroller/default -ojsonpath='{.status.domain}'</pre>
{my-rtas}	<p>The release name of your choice.</p> <p> Note: The release name must consist of alphanumeric characters that are in lowercase or "-" (hyphen). The release name must also start with an alphabetic character and end with an alphanumeric character. For example, <i>my-org</i> or <i>abc-123</i>.</p>
{password-seed}	<p>A value of your choice for the password.</p> <p> Important: This password seed is used to create several default passwords for the server. You must store the password seed securely. When you install the server software by using the backup of the user data, you can reuse the password seed. You can use this seed to restore the backed-up files either on the current or later versions of the server software.</p>
{rlks-ip-address}	<p>The IP address of License Key Server, if you want to set the license value for the first time.</p> <p> Important: When you upgrade the product from the previous version, you must configure the value of License Key Server in the Team Space License Configuration window when the installation of the server is complete.</p>
{my-jaeger-dashboard-url}	<p>The URL of the Jaeger server.</p>

1. Log in to your OCP cluster as a cluster administrator by running the oc login command.
2. Create a namespace in which you want to install the server software by running the following command:

```
oc new-project test-system
```




Remember: The `test-system` is the name of the namespace. If you created a namespace by using a different value, then you must use that value in place of `test-system` in all the instances in this procedure.

3. Perform the following steps to add the Certificate Authority (CA) into a Secret and enable certificates as trusted certificates:

a. Run the following command to verify whether an additional CA is required:

```
curl -sw'{http_code}' -o/dev/null \
"https://wildcard.${oc get -n openshift-ingress-operator ingresscontroller default
-ojsonpath='{.status.domain}'")"
```

If the result of the command is displayed as `503`, the CA is already trusted. You must continue with [4.b.i on page 46](#).

If the result of the command is displayed as `000`, then CA must be added into a Secret. You must continue with [step 3.b on page 45](#).

b. Run the following command to get the default CA in a PEM format:

```
oc get -n openshift-ingress-operator secret router-ca -ojsonpath='{.data.tls.crt}' |
base64 --decode > ca.crt
```

c. Run the following command to validate that the CA used to sign the certificate is the same for ingress:

```
curl -sw'{http_code}' -o/dev/null --cacert ca.crt \
"https://wildcard.${oc get -n openshift-ingress-operator ingresscontroller default
-ojsonpath='{.status.domain}'")"
```

If the result of the command is displayed as `503`, then you must continue with the next step.

If the result of the command is displayed as `000`, then the configuration of the certificate has been customized. You must find the signer of the certificate to continue with the next step.

d. Run the following command to create an ingress Secret to store the CA:

```
oc create secret generic -n test-system ingress --from-file=ca.crt=ca.crt
```

When some components such as static agents or Docker agents want to communicate with Rational® Test Automation Server, the component presents its certificate to the server to verify its identity. Rational® Test Automation Server trusts the component only if it is signed by a recognized and trusted CA. Therefore, you must add the signed CA into a trust by placing it in a Secret to enable certificates as trusted certificates.

4. Perform the following steps to install the server software:

a. Run the following commands to update the `runAsUser` and `fsGroup` to match the Security Context Constraints (SCCs):

```
sed -i -e "s/runAsUser: 1001/runAsUser: $(oc get project test-system -oyaml \
| sed -r -n 's# *openshift.io/sa.scc.uid-range: *([0-9]*)/.*#\1#p')/g;
s/fsGroup: 1001/fsGroup: $(oc get project test-system -oyaml \
```

```
| sed -r -n 's# *openshift.io/sa.scc.supplemental-groups: *([0-9]*)/.*#\1#p')/g"
ibm-rtas-prod/values-openshift.yaml
```

Rational® Test Automation Server is compatible with the restricted SCC. You must run this command to ensure that the runAsUser and fsGroup strategies match with the SCC policy.

- b. Perform one of the steps described in the following table to install the server software based on your requirement:

Step description	Step no
To install the server software	Perform 4.b.i on page 46
To install the server software and enable Jaeger for performance and Web UI tests logs	Perform 4.b.ii on page 46



Remember:

- If you upgrade the product from the previous version, then you must use the same value for `global.persistence.rwxStorageClass` parameter that you used in the previous installation.

You can run the following command to obtain the value that you used for `global.persistence.rwxStorageClass`:

```
oc get pvc -n test-system data-{my-rtas}-userlibs-0 \
-ojsonpath='{.spec.storageClassName}' && echo
```

- The default certificate that terminates TLS connections has a single wildcard. Therefore, you must prefix a single hostname segment for the `global.ibmRtasIngressDomain` parameter.

For example, `--set global.ibmRtasIngressDomain=rtas.{dns name of the openshift cluster}`

- i. Run the following command to install the server software:

```
helm install {my-rtas} ./ibm-rtas-prod -n test-system \
--set license=true \
-f ibm-rtas-prod/values-openshift.yaml \
--set global.persistence.rwxStorageClass=ibmc-file-gold \
--set global.ibmRtasIngressDomain=rtas.{openshift-cluster-dns-name} \
--set global.ibmRtasPasswordAutoGenSeed={password-seed} \
--set global.rationalLicenseKeyServer=@{rlks-ip-address}
```

- ii. Run the following command to install the server software and to enable Jaeger for performance and Web UI tests logs:

```
helm install {my-rtas} ./ibm-rtas-prod -n test-system \
--set license=true \
-f ibm-rtas-prod/values-openshift.yaml \
--set global.persistence.rwxStorageClass=ibmc-file-gold \
--set global.ibmRtasIngressDomain=rtas.{openshift-cluster-dns-name} \
--set global.ibmRtasPasswordAutoGenSeed={password-seed} \
--set global.rationalLicenseKeyServer=@{rlks-ip-address} \
--set-string execution.annotations.sidecar\\.jaegertracing\\.io/inject=true \
--set global.jaegerAgent.internalHostName=localhost \
--set global.jaegerDashboard.externalURL={my-jaeger-dashboard-url}
```

5. **Optional:** Run the following command to remove a job that is used to initialize the PostgreSQL database during the installation of the server software:

```
oc delete job {my-rtas}-postgresql-init -n test-system
```

6. Run the following script from the `ibm-rtas-prod/files` directory to verify and test the installed server software:

```
ibm-rtas-prod/files/helm-test-diag.sh {my-rtas} -n test-system
```

Results

You have installed the server software. The command line displays the following information:

- Keycloak URL to manage and authenticate users.
- A URL to access the Rational® Test Automation Server UI.

What to do next

You can perform the following tasks:

- Configure licenses for team spaces. See [Configuring licenses for team spaces on page 139](#).
- Configure the server software. See [Configuration of the server software on page 105](#).

Related information

[Troubleshooting on page 663](#)

Installation of the server software on Ubuntu

You can find information about the tasks that you can perform to install Rational® Test Automation Server software on the Ubuntu platform.

Prerequisites to install the server software on Ubuntu

You must complete certain tasks before you install Rational® Test Automation Server on the Ubuntu platform.

The following sections describe prerequisites in detail:

- [Internet access on page 48](#)
- [Entitlement key on page 48](#)
- [Ubuntu Server on page 48](#)
- [Backup of user data on page 48](#)
- [Mandatory software on page 49](#)
- [k3s environment on page 49](#)
- [Optional software on page 49](#)
- [Collection of usage metrics on page 50](#)

Internet access

You must have access to the internet to install Rational® Test Automation Server.

Entitlement key

You must copy the **Entitlement key** from the [Container software library](#) to pull the images that are used by the server software. For more information, refer to [Creating an API key](#).

Ubuntu Server

You must install the supported version of Ubuntu Server and have a working Domain Name Server (DNS) that resolves the hostname into a machine-readable IP address.

For more information about specific versions of software requirements, see [System Requirements on page 12](#).



Note: Depending on your testing workload, Rational® Test Automation Server might require more resources. You must use the entire disk space and set up Logical Volume Manager (LVM) by using the `ext4` file system. If your organization requires user data to be stored in a separate partition, then you can create a mount point at `/var/lib/rancher/k3s/storage/` with at least 128 GB capacity.

Backup of user data

If you want to upgrade Rational® Test Automation Server from a previous version, then you must perform one of the following tasks based on the existing version of the server software:

- Back up the data from V10.0.2, Fix Pack 1 or earlier. See [Backing up the user data from a previous release](#).
- Back up the data from V10.1.0 or later. See [Backing up the server data on Ubuntu on page 93](#).



Note: You must select the existing version of Rational® Test Automation Server from the drop-down list to view the procedure to back up and restore the user data for your version because the instructions differ for different versions.

Mandatory software

You must install the following mandatory software:

- Helm. For more information, refer to the [Helm](#) documentation. For more information about specific versions of software requirements, see [System Requirements on page 12](#).



Note: The Helm command must be in one of the directories in your **PATH** environment variable.

- OpenSSH Server. For more information, refer to the Installation section in the [OpenSSH Server](#) documentation.



Note: If you use ssh to log in to the computer, then you can skip this step because OpenSSH is already installed on your computer.

k3s environment

You must set up the Kubernetes environment (k3s) along with other configurations such as firewall, Jaeger, and Prometheus server. See [Setting up a Kubernetes environment \(k3s\) on Ubuntu on page 50](#).

Optional software

You can install the Jaeger operator to trace test logs and Jaeger-based reports when you run tests. For more information, refer to the [Jaeger](#) documentation.

You can then configure Rational® Test Automation Server to use Jaeger traces by passing the following parameters during the installation of the server software:

```
--set global.jaegerDashboard.externalURL={my-jaeger-dashboard-url} \  
--set global.jaegerAgent.internalHostName=localhost
```

Where {my-jaeger-dashboard-url} is the URL of the Jaeger server.

If you use the default values during the installation of Jaeger as suggested in the [Jaeger](#) documentation, then you must pass the following parameters during the installation of the server software:

```
--set global.jaegerDashboard.externalURL=https://$INGRESS_DOMAIN/jaeger \  
--set global.jaegerAgent.internalHostName=jaeger-agent.observability
```

Where \$INGRESS_DOMAIN is the IP address of Jaeger.



Note: You can run the following command to get the value of \$INGRESS_DOMAIN:

```
kubectl get ingress
```

Collection of usage metrics

If you want Rational® Test Automation Server to display the usage statistics of the virtual service instances, you must install the Prometheus server at the time of server installation.

Setting up a Kubernetes environment (k3s) on Ubuntu

You must set up a Kubernetes environment (k3s) so that you can install the server software on the Ubuntu Server. You can use the script that is provided with Rational® Test Automation Server to set up a Kubernetes environment (k3s).

Before you begin

You must have completed the following tasks:

- Installed the following software:
 - OpenSSH server
 - Helm
- Ensured that your computer has a Domain Name Server (DNS) resolvable host name to resolve the host name into a machine-readable IP address.

About this task

As part of the Kubernetes environment (k3s) set up, you can enable the Prometheus server to monitor your system resources by using metrics data.

1. Log in to the Ubuntu server using an SSH session.
2. Run the following command to retrieve the case from the IBM Cloud Pack repository:

```
curl -o ibm-rtas-case-10.1051.0.tgz
https://raw.githubusercontent.com/IBM/cloud-pak/master/repo/case/ibm-rtas-case/10.1051.0/ibm-rtas-case-10.1051.0.tgz
```

3. Run the following commands to fetch the scripts that are used to install Kubernetes:

```
tar xvf ibm-rtas-case-10.1051.0.tgz
chmod +x ibm-rtas-case/inventory/ibmRtasBase/files/*.sh
```

4. Perform one of the steps described in the following table to install the k3s Kubernetes environment based on your requirement:

Step description	Step no
To install the Kubernetes environment (k3s)	Perform 4.a on page 51
To install the Kubernetes environment (k3s) by overriding the default name	Perform 4.b on page 51

- a. Run the following commands to install the Kubernetes with a default domain name that is either based on IP address or fully qualified hostname:

Ubuntu version	Commands to be run
18.04	<pre>\$ cd ibm-rtas-case/inventory/ibmRtasBase/files \$ sudo ./ubuntu-init.sh</pre>
20.04	<pre>\$ cd ibm-rtas-case/inventory/ibmRtasBase/files \$ sudo HOME=\$HOME ./ubuntu-init.sh</pre>

- b. Run the following commands to install the Kubernetes environment (k3s) by overriding the default name:

Ubuntu version	Commands to be run
18.04	<pre>\$ cd ibm-rtas-case/inventory/ibmRtasBase/files \$ sudo INGRESS_DOMAIN={rtas.}myorg.com ./ubuntu-init.sh</pre>
20.04	<pre>\$ cd ibm-rtas-case/inventory/ibmRtasBase/files \$ sudo INGRESS_DOMAIN={rtas.}myorg.com HOME=\$HOME ./ubuntu-init.sh</pre>

Where:

- {rtas.} is a sub-domain name that you specified for the server. For example, `testenv.`



Note: The sub-domain must consist of lowercase alphanumeric characters, "-" (hyphen) or "." (period). Also, the value must start and end with an alphanumeric character.

- myorg.com is the domain name of your organization. For example, `ibm.com`

You can access the product through a web browser by using any of the following methods:

- **Fully Qualified Hostname:** When the server is configured, you can use `hostname -f` command to get the fully qualified host name defined in the DNS to access Rational® Test Automation Server. For example, `rtas.myorg.com`
- **IP address:** You can use the IP address to access Rational® Test Automation Server when you cannot create a specific DNS record for the server. For example, `ip-address.nip.io`.

Result

On completion of the `ubuntu-init.sh` script, a namespace with the name `test-system` is created to install the server software and the output displays the following information on the command-line interface:

- The `INGRESS_DOMAIN` that is in use. This is the URL from where you can access Rational® Test Automation Server. You must use this value for the **global.ibmRtasIngressDomain** parameter in step 5 on page 55 in the server installation topic.
- The DNS information that the Kubernetes cluster uses to resolve names.
- Certificate Authority (CA) that you must import into the browser to prevent certificate errors.

You can run the following command to get the certificate from the system:

```
kubectl get secret ingress -n test-system -o jsonpath={.data.ca\\.crt} | base64 -d
```

- Instructions to confirm whether the Kubernetes environment has started.

You can refer to the **What to do next** section for more details on how to verify that the Kubernetes environment has started.

5. Perform one of the following options to configure a firewall:

Choose from:

- Run the following script to configure the firewall that allows traffic on `cnio` and port `443`:

Ubuntu version	Command to be run
18.04	<code>\$ sudo ./ubuntu-firewall.sh</code>
20.04	<code>\$ sudo HOME=\$HOME ./ubuntu-firewall.sh</code>



Note: You must consult your network administrator before you run this script, and confirm whether the firewall is compatible with your corporate policy.

- Update the firewall that allows traffic on `cnio` and port `443`, if your Ubuntu server is already configured with the firewall.

6. **Optional:** Run the following command to enable the Prometheus server to monitor your system resources by using metrics data:

```
./service.sh expose prometheus
```



Important: The Prometheus metrics are not protected, thus, any information logged into the Prometheus server might be accessible by anyone who has or discovers the `<server-url>/prometheus` URL.

Results

You have set up the Kubernetes environment on Ubuntu.

What to do next

You can perform the following tasks:

- Log in to the server host system again after the installation process is completed so that changes to the group membership are applied.



Note: You can run the `kubectl get pods -A` command to verify that the Kubernetes environment is working. After a while, the status of the pods display a `Running` or `Complete` state.

- Install the server software. See [Installing the server software on Ubuntu by using k3s on page 53](#).

Related information

[Prometheus server](#)

[Troubleshooting on page 663](#)

Installing the server software on Ubuntu by using k3s

You can install Rational® Test Automation Server on the Ubuntu server that has a Kubernetes environment to run functional, integration, and performance tests. Rational® Test Automation Server combines test data, test environments, and test runs and reports into a single, web-based browser for testers and non-testers.



Before you begin



- You must have read and been familiar with additional Helm parameters. See [Additional Helm parameters on page 69](#).
- You must have performed the following tasks:

- Completed the tasks provided in the Prerequisites section. See [Prerequisites to install the server software on Ubuntu on page 47](#).
- Set up the Kubernetes environment (k3s). See [Setting up a Kubernetes environment \(k3s\) on Ubuntu on page 50](#).
- Logged in to the server host system again after you completed the k3s environment setup.
- Copied the following values to be used during the installation process:
 - The **Entitlement key** from the [Container software library](#).
 - The INGRESS_DOMAIN value that is displayed on completion of the `ubuntu-init.sh` script.

About this task

The following table lists the variables that you must replace with the actual value in the command.

Variables	Description
{your_entitlement_key}	The key that you copied from the Container software library .
{example@abc.com}	The email address of the administrator if required.
{my-rtas}	<p>The release name of your choice.</p> <p> Note: The release name must consist of alphanumeric characters that are in lowercase or "-" (hyphen). The release name must also start with an alphabetic character and end with an alphanumeric character. For example, <i>my-org</i> or <i>abc-123</i>.</p>
{my-ingress-dns-name}	The INGRESS_DOMAIN value that is displayed on completion of the <code>ubuntu-init.sh</code> script.
{password-seed}	<p>A value of your choice for the password.</p> <p> Important: This password seed is used to create several default passwords for the server. You must store the password seed securely. When you install the server software by using the backup of the user data, you can reuse the password seed. You</p>

Variables	Description
	 can use this seed to restore the backed-up files either on the current or later versions of the server software.
{rlks-ip-address}	<p>The IP address of License Key Server, if you want to set the license value for the first time.</p>  Important: When you upgrade the product from the previous version, you must configure the value of License Key Server in the Team Space License Configuration window when the installation of the server is complete.

The `test-system` is the name of the namespace that is created during the setting up of the k3s environment.

1. Log in to the Ubuntu server using an SSH session.
2. Add the entitlement registry to Helm to access the server install charts by running the following command:

```
helm repo add ibm-helm https://raw.githubusercontent.com/IBM/charts/master/repo/ibm-helm
```

3. Run the following command to get the latest updates from the repository:

```
helm repo update
```

4. Create a Secret to pull the images that are used by Rational® Test Automation Server by running the following command:

```
kubectl create secret docker-registry cp.icr.io \
-n test-system \
--docker-server=cp.icr.io \
--docker-username=cp \
--docker-password={your_entitlement_key} \
--docker-email={example@abc.com}
```

5. Perform the following steps to install the server software:

- a. Run the following command to get the latest updates from the repository:

```
helm repo update
```

- b. Run the following command to retrieve the charts required to install the server software:

```
helm pull --untar ibm-helm/ibm-rtas-prod --version 10.1051.0
```

- c. Run the following command to provide the execution permission for the scripts that are available in the `files` directory:

```
chmod +x ibm-rtas-prod/files/*.sh
```

- d. Run the following command to install the server software:



Note: To provide enhanced security to Rational® Test Automation Server, the **Sign up** link on the **Login** page of Rational® Test Automation Server is not visible when you install the server software.

If you want to enable the **Sign up** link on the **Login** page, then you must add the `--set keycloak.signup.enabled=true` parameter in the following helm install command.

```
helm install {my-rtas} ./ibm-rtas-prod -n test-system \
--set license=true \
-f ibm-rtas-prod/values-k3s.yaml \
--set global.ibmRtasIngressDomain={my-ingress-dns-name} \
--set global.ibmRtasPasswordAutoGenSeed={password-seed} \
--set global.ibmRtasRegistryPullSecret=cp.icr.io \
--set global.rationalLicenseKeyServer=@{rllks-ip-address}
```

6. **Optional:** Run the following command to remove a job that is used to initialize the PostgreSQL database during the installation of the server software:

```
kubectl delete job {my-rtas}-postgresql-init -n test-system
```

7. Run the following script from the `ibm-rtas-prod/files` directory to verify and test the installed server software:

```
ibm-rtas-prod/files/helm-test-diag.sh {my-rtas} -n test-system
```

Results

On completion of the installation of server software, the output displays the following information on the command-line interface:

- Instructions to access Keycloak to manage and authenticate users.

The username can be *keycloak* and the password can be retrieved by running the following command:

```
kubectl get secret -n test-system rtas-keycloak-postgresql -o jsonpath="{.data.password}" | base64 --decode; echo
```

Where, *rtas* is a sub-domain name that you selected for the server.

- The URL to access the Rational® Test Automation Server UI.

What to do next

You can perform the following tasks:

- Configure licenses for team spaces. See [Configuring licenses for team spaces on page 139](#).
- Configure the server software. See [Configuration of the server software on page 105](#).

You can migrate the user data if you upgraded the server software from the earlier version to a new version.

The migration procedure is different based on the existing version of the server software. For more information about migration process, go to the following topics:

- [Migrating user data on Ubuntu from 10.1.2 or earlier on page 86.](#)
- [Migrating user data on Ubuntu from 10.1.3 or later on page 88.](#)

Related information

[Helm documentation](#)

[Troubleshooting on page 663](#)

Installation of the server software on Azure Kubernetes Service

You can find information about the tasks that you can perform to install Rational® Test Automation Server on the Azure Kubernetes Service (AKS) platform.

Prerequisites for installing the server software on Azure Kubernetes Service

You must complete certain tasks before you install Rational® Test Automation Server on the Azure Kubernetes Service (AKS) platform.

The following sections describe each prerequisite in detail:

- [Internet access on page 57](#)
- [Entitlement key on page 57](#)
- [Azure subscription on page 57](#)
- [Azure Kubernetes Service on page 58](#)
- [Mandatory software on page 58](#)
- [Azure resource provider on page 59](#)

Internet access

You must have access to the internet to install Rational® Test Automation Server.

Entitlement key

You must copy the **Entitlement key** from the [Container software library](#) to pull the images that are used by the server software. For more information, refer to [Creating an API key](#).

Azure subscription

You must have an active Azure subscription along with the following Azure built-in and Azure Active Directory roles:

Categories	Roles	Go to...
Azure built-in	Owner Or Contributor and User Access Administrator	Azure built-in roles in Azure Kubernetes Service documentation.
Azure Active Directory	Application Administrator	Azure AD built-in roles in Azure Kubernetes Service documentation.



Note: You can create a subscription from [Microsoft Azure](#) if you do not have a valid subscription.

Azure Kubernetes Service

You must set up the AKS cluster with Kubernetes.

For more information about specific versions of software requirements, see [System Requirements on page 12](#).

For more information about Azure Kubernetes Service and deployment options, refer to [Azure Kubernetes Service](#) documentation.

Mandatory software

You must install the following mandatory software:

- Git Bash 2.31.1. For more information, refer to the [Git](#) documentation.
- Helm. For more information, refer to the [Helm](#) documentation. For more information about specific versions of software requirements, see [System Requirements on page 12](#).



Note: The Helm command must be in one of the directories in your **PATH** environment variable.

- Azure command-line interface (Azure CLI) 2.34.1. For more information, refer to the [Azure CLI](#) documentation.



Note: You can run the az version command to find the version and dependent libraries that are installed.

- kubectl tool 1.21. For more information, refer to the [Kubernetes](#) documentation.
- Emissary-ingress. See [Installing Emissary-ingress on page 60](#).

- **Optional:** Jaeger operator. For more information, refer to [Jaeger](#) documentation.



Note: You must install Jaeger if you want to trace test logs and Jaeger-based reports after the test run is complete.

Azure resource provider

You must register an Azure resource provider so that the AKS cluster can provision to use Persistent Volumes with Azure Files for the ReadWriteOnceMany (RWX) storage class. You can run the following command to enable the storage for the RWX storage class:

```
az provider register -n Microsoft.Storage
```

For more information about Azure resource providers and types, refer to [Azure Kubernetes Service](#) documentation.

Related information

[Emissary-ingress](#)

Cluster autoscaler on Azure Kubernetes Service

You can enable the cluster autoscaler feature on Azure Kubernetes Service (AKS) to manage the cost of the cluster effectively.

You can use the cluster autoscaler with multiple node pools that are enabled. Depending on your AKS setup, when the nodes are not in use, the nodes of the AKS cluster are automatically shut down to minimize the runtime costs of the AKS cluster without risking the performance of Rational® Test Automation Server.

When you create an AKS cluster, that contains a single node pool as the default node pool. You can use the `az aks node pool` command to add a node pool to your existing AKS cluster. You can then use the `az aks update` command to enable and configure the cluster autoscaler on the node pool for the existing AKS cluster.

You can use the following commands to add a node pool and enable the cluster autoscaler on the node pool for the existing AKS cluster:

```
az aks nodepool add -g <resource_group> -n wrk0 \
  --cluster-name <aks_cluster_name> \
  --kubernetes-version <k8s_version> \
  --node-vm-size Standard_D2ds_v4 \
  --enable-cluster-autoscaler \
  --node-count 1 --min-count 1 --max-count 2 \
  --labels execution=allow \
  --node-taints reserved=reserved:NoSchedule

az aks update \
  -g <resource_group> -n <aks_cluster_name> \
  --cluster-autoscaler-profile scale-down-unnneeded-time=1m scale-down-delay-after-add=1m
```



Note: You must replace `<resource_group>`, `<k8s_version>`, and `<aks_cluster_name>` with the name of the resource group, the version number of Kubernetes, and the name of the AKS cluster in the command.

After you run the commands, you have completed the following actions:

- Added a user node pool called `wrk0`
- Enabled the cluster autoscaler feature
- Updated the default values of **scale-down-unnneeded-time** and **scale-down-delay-after-add** in the cluster-wide autoscaler profile to `1m`

You can add the following command in [Installing the server software on Azure Kubernetes Service on page 64](#) when you install the server software to run test assets on the created node pool:

```
-f rtas-prod/values-dedicated-nodes.yaml
```

After you install the server software by enabling the cluster autoscaler feature on your AKS cluster, a new container is used only to run test assets. You can run the `kubectl get pods` command to view the status of the pod, which is displayed as `Pending`.

When the test asset run is complete, the pod shuts down automatically. Thus, you can reduce the cost of the cluster when the node is not in use. You can then run the `kubectl get nodes` command to verify the number of nodes that are currently in the `Ready` state.

Related information

[Automatically scale a cluster to meet application demands on Azure Kubernetes Service \(AKS\)](#)

[Create and manage multiple node pools for a cluster in Azure Kubernetes Service \(AKS\)](#)

Installing Emissary-ingress

You must install Emissary-ingress to terminate Transport Layer Security (TLS) and route traffic to Rational® Test Automation Server. You can install Emissary-ingress by pulling the images from Azure Container Registry (ACR) when you do not want to pull the images from the public registries.

Before you begin

You must have completed the following tasks:

- Set up the Azure Kubernetes Service (AKS) cluster with Kubernetes. For more information, refer to [Azure Kubernetes Service](#) documentation.
- Installed the following software:

- Git Bash 2.31.1. For more information, refer to the [Git](#) documentation.
- Helm. For more information, refer to the [Helm](#) documentation. For more information about specific versions of software requirements, see [System Requirements on page 12](#).



Note: The Helm command must be in one of the directories in your **PATH** environment variable.

- kubectl tool 1.21. For more information, refer to the [Kubernetes](#) documentation.
- Azure command-line interface (Azure CLI) 2.34.1. For more information, refer to the [Azure CLI](#) documentation.



Note: You can run the az version command to find the version and dependent libraries that are installed.

- Logged in to the AKS cluster.



Note: You can run the az login command to log in to AKS interactively.

- Subscribed to active Azure subscription.



Note: You can run the az account set --subscription {subscription_name} command to set your active subscription.

- Configured the location of your AKS cluster.



Note: You can run the az configure --defaults location={location_name} command to set the location of your AKS cluster.




Tip: You can run the following command to get the list of all the available locations:

```
az account list-locations --query "[].{DisplayName:displayName, Name:name}" -o table
```

About this task

The following table lists the variables that you must replace with the actual value in the command.

Variables	Description
{resource_group}	The name of the resource group that you created during the creation of the AKS cluster.

Variables	Description
{azure_cluster}	The name of the Azure cluster that you created during the creation of the AKS cluster.
{azure_container_registry}	The name of the ACR that you created during the AKS setup.
{ip_address}	The internal IP address to use with the Emissary-ingress.  Note: You must ensure that the IP address is not in use within your virtual network.

1. Run the following command to configure kubectl to connect to the AKS cluster:

```
az aks get-credentials -g {resource_group} -n {azure_cluster}
```



Note: You can verify the connection to the AKS cluster by running the kubectl get nodes command that displays a list of the cluster nodes.

2. Add the Emissary-ingress repository to Helm by running the following command:

```
helm repo add datawire https://app.getambassador.io --force-update
```

3. Run the following command to know the latest version of the Emissary-ingress chart that matches the keyword *emissary-ingress*:

```
helm search repo emissary-ingress
```

4. Run the following command to list the images and move into ACR:

```
for image in $(helm template emissary-ingress --namespace emissary datawire/emissary-ingress \
--set adminService.create=false \
--set agent.enabled=false \
--set replicaCount=1 \
--set test.enabled=false \
--set image.repository=gcr.io/datawire/emissary \
--version "8.1.0" \
| sed -r -n 's/ *image: *([^\"]*)/\1/p' | uniq)
do
  echo "az acr import -n {azure_container_registry} \\"
  echo "  --source ${image}/*?@/@" \\"
  echo "  tag=${image}/@*/"
  echo "  --image ${tag#*/}"
done
```



Remember: emissary-ingress is the name of the namespace. If you created a namespace by using a different value, then you must use that value in place of the emissary-ingress in all the instances in this procedure.

Result

The terminal displays a block of commands.

- Run the commands that are displayed on the terminal after you perform Step 4 on page 62 to import an image to ACR.
- Run the following commands to install the Emissary-ingress by referencing the images from ACR:

```
curl -s https://app.getambassador.io/yaml/emissary/3.1.0/emissary-crds.yaml \
| sed -r -e "s/replicas: 3/replicas: 1/" \
| sed -r -e "s#docker.io/emissaryingress#{azure_container_registry}.azurecr.io/datawire#" \
| kubectl apply -f -

helm install emissary-ingress datawire/emissary-ingress \
-n emissary \
--create-namespace \
--set adminService.create=false \
--set agent.enabled=false \
--set replicaCount=1 \
--set test.enabled=false \
--set image.repository="{azure_container_registry}.azurecr.io/datawire/emissary" \
--set service.annotations."service\.beta\.kubernetes\.io/azure-load-balancer-internal"=true \
--set service.loadBalancerIP={ip_address} \
--version "8.1.0" && \
kubectl rollout status -n emissary deployment/emissary-ingress -w
```

- Run the following command to verify that external-IP is assigned to Emissary-ingress:

```
kubectl get svc -n emissary -w emissary-ingress
```



Note: The status of EXTERNAL-IP is displayed as `Pending` for a few minutes. If the status does not change to `{ip_address}`, then that indicates you have a permission issue.

You can run the following command to investigate the issue:

```
kubectl describe svc -n emissary emissary-ingress
```

Results

You have installed Emissary-ingress.

What to do next

You can install the server software. See [Installing the server software on Azure Kubernetes Service on page 64](#).

Related information

[Create an ingress controller to an internal virtual network in Azure Kubernetes Service \(AKS\)](#)

[Create an ingress controller to an internal virtual network in Azure Kubernetes Service \(AKS\)](#)

[Emissary-ingress documentation](#)

Installing the server software on Azure Kubernetes Service

You can install Rational® Test Automation Server on Azure Kubernetes Service (AKS) that has a Kubernetes environment to run functional, integration, and performance tests. Rational® Test Automation Server combines all the capabilities into a single web-based browser for testers and non-testers. The capabilities include test runs, test data, test environment, or test reports.

Before you begin

- You must have read and been familiar with additional Helm parameters. See [Additional Helm parameters on page 69](#).
- You must have performed the following tasks:
 - Completed the tasks provided in the Prerequisites section. See [Prerequisites for installing the server software on Azure Kubernetes Service on page 57](#).
 - **Optional:** Enabled the autoscaler feature if you want a new container to be used only to run test assets. See [Cluster autoscaler on Azure Kubernetes Service on page 59](#).
 - Copied the **Entitlement key** from the [Container software library](#).
 - Logged in to the AKS cluster.



Note: You can run the az login command to log in to AKS interactively.

- Subscribed to active Azure subscription.



Note: You can run the az account set --subscription {subscription_name} command to set your active subscription.

- Configured the location of your AKS cluster.



Note: You can run the `az configure --defaults location={location_name}` command to set the location of your AKS cluster.





Tip: You can run the following command to get the list of all the available locations:

```
az account list-locations --query "[].{DisplayName:displayName, Name:name}" -o table
```

About this task

The following table lists the variables that you must replace with the actual value in the command.

Variables	Description
{resource_group}	The name of the resource group that you created during the creation of the AKS cluster.
{azure_cluster}	The name of the Azure cluster that you created during the creation of the AKS cluster.
{azure_container_registry}	The name of the ACR that you created during the AKS setup.
{my-ingress-dns-name}	The Ingress name and DNS zone value that you provided during the setting up of the AKS cluster. For example, if the Ingress name and DNS zone value that you provided are <code>rationaltest</code> , <code>my-org.nonprod</code> , then <code>{my-ingress-dns-name}</code> value must be <code>rationaltest.myorg.nonprod</code> .
{your_entitlement_key}	The key that you copied from the Container software library .
{my-rtas}	The release name of your choice. Note: The release name must consist of alphanumeric characters that are in lowercase or "-" (hyphen). The release name must also start with an alphabetic character and end with an alphanumeric character. For example, <code>my-org</code> or <code>abc-123</code> .
{rlks-ip-address}	The IP address of License Key Server, if you want to set the license value for the first time.

Variables	Description
	 Important: When you upgrade the product from the previous version, you must configure the value of License Key Server in the Team Space License Configuration window when the installation of the server is complete.
{password-seed}	<p>A value of your choice for the password.</p>  Important: This password seed is used to create several default passwords for the server. You must store the password seed securely. When you install the server software by using the backup of the user data, you can reuse the password seed. You can use this seed to restore the backed-up files either on the current or later versions of the server software.
{ip_address}.nip.io	<p>The IP address that is assigned to Emissary-ingress.</p> <p>You can run the following command to obtain the IP address of Emissary-ingress:</p> <pre>kubectrl get ingress -A</pre>



Remember: The `test-system` is the name of the namespace. If you created a namespace by using a different value, then you must use that value in place of `test-system` in all the instances in this procedure.

1. Add the entitlement registry to Helm by running the following command:

```
helm repo add ibm-helm https://raw.githubusercontent.com/IBM/charts/master/repo/ibm-helm
```

2. Run the following command to get the latest updates from the repository:

```
helm repo update
```

3. Run the following commands to fetch the scripts that are used to install Rational® Test Automation Server:

```
helm pull --untar ibm-helm/ibm-rtas-prod --version 10.1051.0
```

4. Run the following command to configure kubectl to connect to the AKS cluster:


```
az aks get-credentials -g {resource_group} -n {azure_cluster}
```



Note: You can verify the connection to the AKS cluster by running the `kubectl get nodes` command that displays a list of the cluster nodes.

5. Run the following script from the `ibm-rtas-prod/files` directory to pull the images of Rational® Test Automation Server from the entitlement registry to ACR:

```
PULL_ARGUMENTS="-u cp -p {your_entitlement_key}" \  
bash ibm-rtas-prod/files/move-images.sh $REGISTRY.azurecr.io/ibm-rtas cp.icr.io/cp
```

 **Tip:** You can verify that multiple manifests are associated with the image in the ACR by running the following commands:

```
az acr repository list -n {azure_container_registry} -otsv
az acr repository show-manifests -n {azure_container_registry} --repository \
$(az acr repository list -n {azure_container_registry} -otsv --query [0])
```

6. Create a namespace in which you want to install the server software by running the following command:

```
kubectl create namespace test-system
```

7. Perform one of the following steps to enable Certificate Authority (CA) as trusted certificates and to create an ingress Secret:

a. Run the following script from the `ibm-rtas-prod/files` directory if your endpoints can resolve Azure private DNS zones:

```
ibm-rtas-prod/files/certificate.sh -n test-system -s ingress {my-ingress-dns-name}
```

b. Run the following script from the `ibm-rtas-prod/files` directory if your endpoints cannot resolve Azure private DNS zones:

```
ibm-rtas-prod/files/certificate.sh -n test-system -s ingress {ip_address}.nip.io
```

8. Run the following command to install the server software:

The following command installs the server software by referencing the images from ACR. Thereby, Rational® Test Automation Server can provide better performance.



Notes:

- If you enabled the autoscaler feature to use a new container only to run test assets, then you must add `-f rtas-prod/values-dedicated-nodes.yaml` in the following helm install command.
- To provide enhanced security to Rational® Test Automation Server, the **Sign up** link on the **Login** page of Rational® Test Automation Server is not visible when you install the server software.

If you want to enable the **Sign up** link on the **Login** page, then you must add the `--set keycloak.signup.enabled=true` parameter in the following helm install command.

```
helm install {my-rtas} ./ibm-rtas-prod -n test-system \
--set license=true \
-f ibm-rtas-prod/values-k8s.yaml \
--set global.persistence.rwxStorageClass=azurefile \
--set global.ibmRtasIngressDomain={my-ingress-dns-name} \
--set global.ibmRtasRegistry=<azure_container_registry>.azurecr.io/ibm-rtas \
--set global.ibmRtasPasswordAutoGenSeed={password-seed} \
--set global.rationalLicenseKeyServer=@{rlks-ip-address}
```

9. Run the following script from the `ibm-rtas-prod/files` directory to verify and test the installed server software:

```
ibm-rtas-prod/files/helm-test-diag.sh {my-rtas} -n test-system
```

Results

You have installed the server software. The terminal displays the following information:

- The URL to access Keycloak to manage and authenticate users.

The username can be `keycloak` and the password can be retrieved by running the following command:

```
kubectl get secret -n test-system {my-rtas}-keycloak-postgresql -o jsonpath="{.data.password}" | base64 --decode; echo
```

Where:

- `{my-rtas}` is the name of the release that was provided during the installation of the server software.
- `test-system` is the name of the namespace that you created during the installation of the server software.
- The URL to access the Rational® Test Automation Server UI.

What to do next

You can perform the following tasks:

- Configure licenses for team spaces. See [Configuring licenses for team spaces on page 139](#).
- Configure the server software. See [Configuration of the server software on page 105](#).

Related information

[Helm documentation](#)

[Azure Kubernetes Service documentation](#)

Management of server features

After you install the server software, you can manage certain features based on your requirements.

During the installation of the server software, if you did not use any Helm parameters, you can still use those parameters even after installation is complete.

You can use the `helm upgrade` command to enable or disable server features after the installation of Rational® Test Automation Server is complete.

For example, if you want to enable the Egress policy that restricts traffic to private IP addresses by retaining the values of other parameters that you used during the installation, you can run the following command after the installation is complete:


```
helm upgrade {my-rtas} -n {namespace} ./ibm-rtas-prod \
--reuse-values \
--set networkPolicy.egress.enabled=true
```




Note: You must substitute {my-rtas} and {namespace} with the name of the release and namespace that you provided during the installation of the server software.


Similarly, you can use the other additional Helm parameters to enable or disable features based on your requirements. See [Additional Helm parameters on page 69](#).


Additional Helm parameters

You can find the information about Helm parameters that you can use during the installation of Rational® Test Automation Server.

As a system administrator when you install or upgrade Rational® Test Automation Server, you can pass values for some of the individual parameters by using `--set`. The following table lists such parameters that you can use while running the `helm install` or `helm upgrade` command to enable or disable certain features in Rational® Test Automation Server.

Parameters	Values	Description
keycloak.signup-enabled	true false	<p>Use this parameter to show or hide the Sign-Up link on the Rational® Test Automation Server Login page.</p> <p>The default value of the <code>keycloak.signup.enabled</code> parameter is set to <code>false</code> to enhance the security of Rational® Test Automation Server.</p> <p> Note: You can use this parameter only during the installation of the server software.</p> <p>If you have administrator access to Keycloak, then you can also enable the self-registration by using the Keycloak UI after the installation of Rational® Test Automation Server is complete. For more information, refer to the Keycloak documentation.</p>
gateway.allowed-Origin	IP address or domain name of other servers.	<p>Use this parameter to integrate call requests from other web servers to Rational® Test Automation Server.</p> <p>The default configuration of Rational® Test Automation Server rejects all Cross-origin resource sharing (CORS). Therefore, you must use the <code>gateway.allowedOrigin</code> parameter to restrict CORS to a specific whitelist.</p> <p>For example,</p>

Parameters	Values	Description
		<pre data-bbox="630 262 1299 346">--set gateway.allowedOrigin=http://http.example.com:*,https:// https.mydomain.net:*</pre> <p data-bbox="613 367 1388 436">Where, <code>http.example.com</code> and <code>https.mydomain.net</code> are the domains that you want to provide access to Rational® Test Automation Server.</p> <p data-bbox="613 472 1404 625">You can also provide the value of <code>gateway.allowedOrigin</code> as the IP address when you cannot resolve a specific Domain Name Server (DNS). If your domain can be accessible through IP address on a specific port, then you can provide the value of <code>gateway.allowedOrigin</code> as follows:</p> <pre data-bbox="630 661 1404 714">--set gateway.allowedOrigin=*/ip-address:*,*/ip-address:port_number</pre> <p data-bbox="613 745 755 777">For example,</p> <pre data-bbox="630 798 1153 829">--set gateway.allowedOrigin="*/192.0.2.0:*</pre> <pre data-bbox="630 850 1193 882">--set gateway.allowedOrigin="*/192.0.2.1:3000"</pre> <p data-bbox="630 913 1404 997"> Note: You can provide multiple domain names or IP addresses separated by a comma (,).</p>
networkPolicy.egress.enabled	true false	<p data-bbox="613 1081 1388 1155">Set this parameter value to <code>true</code> to restrict traffic from Rational® Test Automation Server to private IP addresses.</p> <p data-bbox="613 1186 1421 1386">The default configuration of Rational® Test Automation Server does not create any Egress policy to restrict the other endpoints that can connect with Rational® Test Automation Server. Therefore, you must use the <code>networkPolicy.egress.enabled</code> parameter to enable the Egress policy that can restrict traffic to private IP addresses.</p> <p data-bbox="613 1417 755 1449">For example,</p> <pre data-bbox="630 1470 1088 1501">--set networkPolicy.egress.enabled=true</pre> <p data-bbox="613 1533 1372 1564">You can inspect the deployed policy by running the following command:</p> <pre data-bbox="630 1585 1372 1638">kubectl get networkpolicy -n {namespace} {my-rtas}-allow-egress -oyaml</pre>

Parameters	Values	Description
		 Note: You must substitute <code>{namespace}</code> with the name of the namespace and <code>{my-rtas}</code> with the release name that you used during the installation of the server software.
execution.template. resources.limits. memory	<code><size_of_memory>Gi</code>	<p>Use this parameter to run VU schedules that contain multiple Web UI tests or a large number of virtual users or a combination of Web UI, performance, and API tests.</p> <p>The default value of the <code>execution.template.resources.limits.memory</code> parameter is set to <code>3Gi</code>.</p> <p>For example,</p> <pre>--set execution.template.resources.limits.memory=5Gi</pre>
global. ibmRtasCertSecretOptional	<code>true</code> <code>false</code>	<p>Set this parameter value to <code>false</code> if you create an ingress Secret to store certificates as trusted certificates.</p> <p>The default value of the <code>global.ibmRtasCertSecretOptional</code> parameter is set to <code>true</code>.</p> <p>For example,</p> <pre>--set global.ibmRtasCertSecretOptional=false</pre>

Related information

[Installing the server software on IBM Red Hat OpenShift on page 33](#)

[Installing the server software on Ubuntu by using k3s on page 53](#)

[Installing the server software on Azure Kubernetes Service on page 64](#)


[Management of server features on page 68](#)

Server software upgrade methods

When you want to use the enhanced functionalities of Rational® Test Automation Server, you must upgrade Rational® Test Automation Server to the latest version.

You can upgrade the server software by using following methods:

- **In-place upgrade:** In this method, you can upgrade the server software in the same location that contained 10.1.3 or later version of the server software by using the helm upgrade command.
- **Upgrade (by new install):** In this method, you can upgrade the server software by uninstalling the existing version of Rational® Test Automation Server, and then by installing the latest version of the server software.

 **Important:** On the Ubuntu platform, the **In-place upgrade** method is not available. You must use the **Upgrade (by new install)** method to upgrade the server software.

The upgrade procedures are different based on the existing version of the software. To learn more about server software upgrade methods on platforms such as Red Hat OpenShift, Azure Kubernetes Service (AKS) and Ubuntu, refer to the following topics:

Server software upgrade on Red Hat OpenShift

When you want to use the latest version of the server software on Red Hat OpenShift, you must upgrade Rational® Test Automation Server from the previous version of the software.

On the Red Hat OpenShift platform, you can either use **In-place upgrade** or **Upgrade (by new install)** methods depending on existing version of the server software.

You can refer to the following table to know the upgrade method that you want to use:

Existing version of the software	Method to use
10.1.3 or later	In-place upgrade
10.1.2 or earlier	Upgrade (by new install)

Upgrading the server software by using the in-place upgrade method

When you want to install the latest version of the server software in the same location that contained the 10.1.3 or later, then you can upgrade Rational® Test Automation Server by using the helm upgrade command.

Before you begin

- You must have read and been familiar with additional Helm parameters. See [Additional Helm parameters on page 69](#).
- You must have completed the following tasks:

- Informed users that Rational® Test Automation Server is offline temporarily during the upgrade process.
- Installed Helm. For more information, refer to the Installing a Helm chart on an Red Hat OpenShift Container Platform cluster section in the [Red Hat OpenShift](#) documentation.
- Completed all test executions that are running on the existing version of Rational® Test Automation Server.
- Stopped all stub executions that are running on the existing version of Rational® Test Automation Server.
- Canceled any scheduled test runs that have a future date or time.
- Copied an Entitlement key from the [Container software library](#) to pull the images used by the server software. For more information, refer to [Creating an API key](#).

Optional. You can back up the user data from the previous version of the product. If the upgrade fails, then you can use that backup file to restore it on latest version of the server software. See [Backing up the server data on Red Hat OpenShift on page 91](#).

For more information about specific versions of software requirements, see [System Requirements on page 12](#).

About this task

The following table lists the variables that you must replace with the actual value in the command.

Variables	Description
{your_entitlement_key}	The key that you copied from the Container software library .
{example@abc.com}	The email address of the administrator if required.
{my-rtas}	The release name that you provided during the installation of the server software.
{openshift-cluster-dns-name}	The ingress DNS name that you selected for the server. You can run the following command to obtain the default value of {openshift-cluster-dns-name}: <pre>oc get --namespace=openshift-ingress-operator ingresscontroller/default -jsonpath='{.status.domain}'</pre>

Variables	Description
{password-seed}	The password that you provided during the installation of the server software.



Tip: You can run the following command to know the value of the variables that you used during the installation of the previous version of the server software:

```
helm get values {release_name} -n {namespace}
```

For example, if your name of the release is `my-rtas` and the name of the namespace is `testsystem`, then you must run the command as follows:

```
helm get values my-rtas -n testsystem
```

1. Log in to your OCP cluster as a cluster administrator by running the `oc login` command.
2. Add the entitlement registry to Helm to access the server install charts by running the following command:

```
helm repo add ibm-helm https://raw.githubusercontent.com/IBM/charts/master/repo/ibm-helm
```

3. Run the following command to get the latest updates from the repository:

```
helm repo update
```

4. Run the following command to retrieve the charts:

```
helm pull --untar ibm-helm/ibm-rtas-prod --version 10.1051.0
```

5. Create a Secret to pull images that are used by Rational® Test Automation Server by running the following commands:

```
oc create secret docker-registry cp.icr.io \
-n test-system \
--docker-server=cp.icr.io \
--docker-username=cp \
--docker-password={your_entitlement_key} \
--docker-email={example@abc.com}
```

6. Perform the following steps to upgrade the server software:

- a. Run the following commands to update the `runAsUser` and `fsGroup` to match the Security Context Constraints (SCC):

```
sed -i -e "s/runAsUser: 1001/runAsUser: $(oc get project test-system -oyaml \
| sed -r -n 's# *openshift.io/sa.scc.uid-range: *([0-9]*)/.*#\1#p')/g;
s/fsGroup: 1001/fsGroup: $(oc get project test-system -oyaml \
| sed -r -n 's# *openshift.io/sa.scc.supplemental-groups: *([0-9]*)/.*#\1#p')/g"
ibm-rtas-prod/values-openshift.yaml
```

Rational® Test Automation Server is compatible with the restricted Security Context Constraint. You must run this command to ensure that the `runAsUser` and `fsGroup` strategies match with the SCC policy.

- b. Perform one of the steps described in the following table to upgrade the server software based on your requirement:

Step description	Step no
To upgrade the server software	Perform 6.b.i on page 75
To upgrade the server software and enable Jaeger for performance and Web UI tests logs	Perform 6.b.ii on page 75



Remember:

- You must use the same value for the `global.persistence.rwxStorageClass` parameter that you used in the previous installation.

You can run the following command to obtain the value that you used for

`global.persistence.rwxStorageClass`:

```
oc get pvc -n test-system data-{my-rtas}-userlibs-0 \
  -ojsonpath='{.spec.storageClassName}' && echo
```

- The default certificate that terminates TLS connections has a single wildcard. Therefore, you must prefix a single hostname segment for the `global.ibmRtasIngressDomain` parameter.

For example, `--set global.ibmRtasIngressDomain=rtas.{dns name of the openshift cluster}`

- i. Run the following command to upgrade the server software:

```
helm upgrade {my-rtas} ./ibm-rtas-prod -n test-system \
--set license=true \
-f ibm-rtas-prod/values-openshift.yaml \
--set global.persistence.rwxStorageClass=ibmc-file-gold \
--set global.ibmRtasIngressDomain=rtas.{openshift-cluster-dns-name} \
--set global.ibmRtasPasswordAutoGenSeed={password-seed} \
--set global.ibmRtasRegistryPullSecret=cp.icr.io
```

- ii. Run the following command to upgrade the server software and to enable Jaeger for performance and Web UI tests logs:

```
helm upgrade {my-rtas} ./ibm-rtas-prod -n test-system \
--set license=true \
-f ibm-rtas-prod/values-openshift.yaml \
--set global.persistence.rwxStorageClass=ibmc-file-gold \
--set global.ibmRtasIngressDomain=rtas.{openshift-cluster-dns-name} \
--set global.ibmRtasPasswordAutoGenSeed={password-seed} \
--set global.ibmRtasRegistryPullSecret=cp.icr.io \
```

```
--set-string execution.annotations.sidecar\\.jaegertracing\\.io/inject=true \
--set global.jaegerAgent.internalHostName=localhost \
--set global.jaegerDashboard.externalURL={my-jaeger-dashboard-url}
```

7. Run the following command to verify and test the upgraded server software:

```
ibm-rtas-prod/files/helm-test-diag.sh {my-rtas} -n test-system
```

Results

On the successful upgrade of Rational® Test Automation Server, the output displays the following information:

- Keycloak URL to manage and authenticate users.
- A URL to access the Rational® Test Automation Server UI.

What to do next

- Configure licenses for team spaces. See [Configuring licenses for team spaces on page 139](#).
- Configure the server software. See [Configuration of the server software on page 105](#).

Related information

[Troubleshooting on page 663](#)

Server software upgrade by using upgrade (by new install) method

When your Rational® Test Automation Server is 10.1.2 or earlier, then to upgrade the server software, you must uninstall the existing version of the server software and install the latest version.

Before you upgrade the server software, you must ensure that you have completed the following tasks:

- Installed Helm. For more information, refer to the Installing a Helm chart on an Red Hat OpenShift Container Platform cluster section in the [Red Hat OpenShift](#) documentation.
- Completed all test executions that are running on the existing version of Rational® Test Automation Server.
- Stopped all stub executions that are running on the existing version of Rational® Test Automation Server.
- Canceled any scheduled test runs that have a future date or time.
- Informed users that Rational® Test Automation Server is offline temporarily during the upgrade process.

For more information about specific versions of software requirements, see [System Requirements on page 12](#).

In upgrade (by new install) method, depending on the existing version of the software, you can upgrade the server software by using any of the following options.

To upgrade Rational® Test Automation Server (10.1.2 or earlier) to latest version, you must perform these tasks in sequence as listed in the following table:

Tasks	Task description	More information
1	Back up the user data from the previous version of Rational® Test Automation Server.	Backing up the server data on Red Hat OpenShift on page 91
2	Uninstall the previous version of Rational® Test Automation Server.	Uninstalling the server software from IBM Red Hat OpenShift on page 102
3	Install the latest version of the server software.	Installing the server software on IBM Red Hat OpenShift on page 33
4	Migrate the user data after the installation is complete.	Migrating user data on Red Hat OpenShift on page 85

If your existing version of server software is 10.0.2, Fix Pack 1, or earlier, then you cannot directly upgrade to the latest version of the server software due to the steps involved in the migration process. Therefore, you must perform the tasks in sequence as listed in the following table:

Tasks	Task description	More information
1	Upgrade Rational® Test Automation Server 10.1.2.	Installation of the server software on IBM Red Hat OpenShift on page 30
2	Back up the user data from Rational® Test Automation Server 10.1.2.	Backing up the server data on Red Hat OpenShift on page 91
3	Uninstall Rational® Test Automation Server 10.1.2.	Uninstalling the server software on page 102
4	Install the latest version of the server software.	Installing the server software on IBM Red Hat OpenShift on page 33
5	Migrate the user data after the installation is complete.	Migrating user data on Red Hat OpenShift on page 85

Server software upgrade on Ubuntu

When you want to use the latest version of the server software on Ubuntu, you must upgrade Rational® Test Automation Server from the previous version of the software.

Before you upgrade the server software, you must ensure that you have completed the following tasks:

- Installed Helm. For more information, refer to the [Helm](#) documentation.
- Completed all test executions that are running on the existing version of Rational® Test Automation Server.
- Stopped all stub executions that are running on the existing version of Rational® Test Automation Server.

- Canceled any scheduled test runs that have a future date or time.
- Informed users that Rational® Test Automation Server is offline temporarily during the upgrade process.

For more information about specific versions of software requirements, see [System Requirements on page 12](#).

In upgrade (by new install) method, depending on the existing version of the software, you can upgrade the server software by using any of the following options.

To upgrade Rational® Test Automation Server (10.1.0 or later) to latest version, you must perform these tasks in sequence as listed in the following table:

Tasks	Task description	More information
1	Back up the user data from the previous version of Rational® Test Automation Server.	Backing up user data on page 93
2	Uninstall the previous version of Rational® Test Automation Server including entire Kubernetes environment.	Uninstalling the server software from Ubuntu on page 103
3	Set up the Kubernetes environment (k3s) by using the provided script.	Setting up a Kubernetes environment (k3s) on Ubuntu on page 50
4	Install the latest version of the server software.	Installing the server software on Ubuntu by using k3s on page 53
5	Migrate the user data after the installation is complete.	Migrating user data on Ubuntu from 10.1.2 or earlier on page 86 Or Migrating user data on Ubuntu from 10.1.3 or later on page 88

If your existing version of server software is 10.0.2, Fix Pack 1, or earlier, then you cannot directly upgrade to the latest version of the server software due to the steps involved in the migration process. Therefore, you must perform the tasks in sequence as listed in the following table:

Tasks	Task description	More information
1	Upgrade Rational® Test Automation Server 10.1.2.	Installation of the server software on Ubuntu on page 47
2	Back up the user data from Rational® Test Automation Server 10.1.2.	Backing up user data - 10.1.2 on page 93
3	Uninstall Rational® Test Automation Server 10.1.2 including entire Kubernetes environment.	Uninstalling the server software on page 103

Tasks	Task description	More information
4	Install the latest version of the server software.	Installing the server software on Ubuntu by using k3s on page 53
5	Migrate the user data after the installation is complete.	Migrating user data on Ubuntu from 10.1.2 or earlier on page 86

Upgrading the server software on Azure Kubernetes Service

When you want to use the latest version of the server software on Azure Kubernetes Service (AKS), you must upgrade Rational® Test Automation Server from the previous version of the software.

Before you begin

- You must have read and been familiar with additional Helm parameters. See [Additional Helm parameters on page 69](#).
- You must have completed the following tasks:
 - Completed the tasks provided in the Prerequisites section. See [Prerequisites for installing the server software on Azure Kubernetes Service on page 57](#).
 - Installed Emissary-ingress. See [Installing Emissary-ingress on page 60](#).
 - Informed users that Rational® Test Automation Server is offline temporarily during the upgrade process.
 - Completed all test executions that are running on the existing version of Rational® Test Automation Server.
 - Stopped all stub executions that are running on the existing version of Rational® Test Automation Server.
 - Canceled any scheduled test runs that have a future date or time.
 - Logged in to the AKS cluster.



Note: You can run the az login command to log in to AKS interactively.

- Subscribed to active Azure subscription.



Note: You can run the az account set --subscription {subscription_name} command to set your active subscription.

Optional. You can back up the user data from the earlier version of the product. If the upgrade fails, then you can use that backup file to restore it to the latest version of the server software. See [Backing up the server data on Azure Kubernetes Service on page 100](#).



Note: For more information about specific versions of software requirements, see [System Requirements on page 12](#).

About this task

When you want to install the latest version of the server software in the same location that contained the 10.2.3 or later, then you can upgrade Rational® Test Automation Server by using the helm upgrade command.

The following table lists the variables that you must replace with the actual value in the command.

Variables	Description
{resource_group}	The name of the resource group that you created during the creation of the AKS cluster.
{azure_cluster}	The name of the Azure cluster that you created during the creation of the AKS cluster.
{azure_container_registry}	The name of the ACR that you created during the AKS setup.
{my-ingress-dns-name}	The Ingress name and DNS zone value that you provided during the setting up of the AKS cluster. For example, if the Ingress name and DNS zone value that you provided are <code>rationaltest</code> , <code>my-org.nonprod</code> , then <code>{my-ingress-dns-name}</code> value must be <code>rationaltest.myorg.nonprod</code> .
{my-rtas}	The release name that you provided during the installation of the server software.



Remember: The `test-system` is the name of the namespace that you created during the installation of the server software. If you created a namespace by using a different value, then you must use that value in place of `test-system` in all the instances in this procedure.



Tip: You can run the following command to know the value of the variables that you used during the installation of the previous version of the server software:

```
helm get values {release_name} -n {namespace}
```

For example, if your name of the release is `my-rtas` and the name of the namespace is `testsystem`, then you must run the command as follows:

```
helm get values my-rtas -n testsystem
```

1. Add the entitlement registry to Helm by running the following command:

```
helm repo add ibm-helm https://raw.githubusercontent.com/IBM/charts/master/repo/ibm-helm
```

2. Run the following command to get the latest updates from the repository:

```
helm repo update
```

3. Run the following commands to fetch the scripts that are used to install Rational® Test Automation Server:

```
helm pull --untar ibm-helm/ibm-rtas-prod --version 10.1051.0
```

4. Run the following command to configure kubectl to connect to the AKS cluster:

```
az aks get-credentials -g {resource_group} -n {azure_cluster}
```



Note: You can verify the connection to the AKS cluster by running the `kubectl get nodes` command that displays a list of the cluster nodes.

5. Run the following script from the `ibm-rtas-prod/files` directory to pull the images of Rational® Test Automation Server from the entitlement registry to ACR:

```
PULL_ARGUMENTS="-u cp -p {your_entitlement_key}" \
  bash ibm-rtas-prod/files/move-images.sh $REGISTRY.azurecr.io/ibm-rtas cp.icr.io/cp
```



Tip: You can verify that multiple manifests are associated with the image in the ACR by running the following commands:

```
az acr repository list -n {azure_container_registry} -otsv
az acr repository show-manifests -n {azure_container_registry} --repository \
$(az acr repository list -n {azure_container_registry} -otsv --query [0])
```

6. Run the `helm list -A` command to list all releases that are deployed across all namespaces.
7. Run the following command to upgrade the server software:

```
helm upgrade {my-rtas} ./ibm-rtas-prod -n test-system \
--set license=true \
```

```
-f ibm-rtas-prod/values-k8s.yaml \
--set global.persistence.rwxStorageClass=azurefile \
--set global.ibmRtasIngressDomain={my-ingress-dns-name} \
--set global.ibmRtasRegistry=<azure_container_registry>.azurecr.io/ibm-rtas \
--set global.ibmRtasPasswordAutoGenSeed={password-seed}
```



Note: The following command upgrades the server software by referencing the images from ACR. Therefore, Rational® Test Automation Server can provide better performance.

- Run the following script from the `ibm-rtas-prod/files` directory to verify and test the installed server software:

```
ibm-rtas-prod/files/helm-test-diag.sh {my-rtas} -n test-system
```

Results

On the successful upgrade of Rational® Test Automation Server, the output displays the following information:

- Keycloak URL to manage and authenticate users.
- A URL to access the Rational® Test Automation Server UI.

What to do next

- Configure licenses for team spaces. See [Configuring licenses for team spaces on page 139](#).
- Configure the server software. See [Configuration of the server software on page 105](#).

Migration of user data

You can find information about the migration limitations and methods when you want to migrate data such as projects, test resources, configuration details of integrated applications, and so on. You can migrate data that is generated in an earlier version of IBM® Rational® Test Automation Server after you upgrade to or install the latest version, and then use that data in the latest version.

Limitations for user data migration

Before you can decide on the migration method for the version of Rational® Test Automation Server, you must review the limitations that apply to the migrated data.

Table 2. Migration limitations

Migrating from	Migrating to	Limitation	The workaround, if available
10.5.0	10.5.1	You cannot migrate the IBM® Engineering Test Management data that is generated when it is configured as a quality-	You must reconfigure IBM® Engineering Test Management as a quality man-

Table 2. Migration limitations (continued)

Migrating from	Migrating to	Limitation	The workaround, if available
		ty management solution on Rational® Test Automation Server 10.5.0 to Rational® Test Automation Server 10.5.1.	agement solution in the latest version of Rational® Test Automation Server.
10.5.0 or earlier	10.5.1	If you configured Jira as a change management system on Rational® Test Automation Server, and subsequently changed the name of the project in Jira, then you cannot then configure the changed project name in the CHANGE MANAGEMENT SYSTEM tab of Rational® Test Automation Server 10.5.1.	You must reconfigure Jira as a change management system in the latest version of Rational® Test Automation Server.
10.5.0 or earlier	10.5.1	If you configured IBM® Engineering Test Management as a quality management solution on Rational® Test Automation Server 10.5.0, and subsequently updated or deleted a Project area in Engineering Test Management, then you cannot perform the following tasks: <ul style="list-style-type: none"> • Delete your project from Rational® Test Automation Server. • Delete the configured Engineering Test Management from the Rational® Test Automation Server project. 	You must reconfigure IBM® Engineering Test Management as a quality management solution in the latest version of Rational® Test Automation Server.
10.2.3	10.5.0	If you configured IBM® Engineering Test Management on Rational® Test Automation Server 10.2.3, the information about Engineering Test Management is not visible after you migrate Rational® Test Automation Server from 10.2.3 to 10.5.0 or later.	You must reconfigure IBM® Engineering Test Management as a quality management solution in the latest version of Rational® Test Automation Server.

Table 2. Migration limitations (continued)

Migrating from	Migrating to	Limitation	The workaround, if available
10.2.3	10.5.0	If you configured Jira on Rational® Test Automation Server 10.2.3, the information about Jira is not visible after you migrate Rational® Test Automation Server from 10.2.3 to 10.5.0 or later.	You must reconfigure Jira as a change management system in the latest version of Rational® Test Automation Server.
10.2.3 or earlier	10.5.0	You cannot migrate edited test assets that were created in Rational® Test Automation Server 10.2.3 or earlier, to Rational® Test Automation Server 10.5.0 or later.	You might have to open and edit the test assets in the latest version of Rational® Test Automation Server.
10.2.0	10.2.1 or later	If you edited test assets in your project on Rational® Test Automation Server 10.2.0 and migrated them, then the values displayed in the Last Changed By field might be incorrect for the edited test assets.	

Migration methods

The data that is generated in Rational® Test Automation Server can be migrated and used in a later version of Rational® Test Automation Server with except for the limitations detailed in [Table 2: Migration limitations on page 82](#).

You can refer to the following table to know the details of the migration when you want to upgrade to the latest Rational® Test Automation Server version:

Table 3. Migration method used during an upgrade

OS / Platform	Server version	Upgrade method	Migration method
Red Hat OpenShift	10.1.3 or later	In-place upgrade	When you upgrade the software by using the helm upgrade command, the migration of user data takes place during the upgrade process.

Table 3. Migration method used during an upgrade (continued)

OS / Platform	Server version	Upgrade method	Migration method
	10.1.2 or earlier	Upgrade by new install	<p>When you upgrade the server software by uninstalling the existing version, and installing the required version of Rational® Test Automation Server, then you must perform the migration task after the installation of the server software is complete.</p> <p>For more information about the migration process, see Migrating user data on Red Hat OpenShift on page 85.</p>
Ubuntu	10.1.3 or later	Upgrade by new install	<p>When you upgrade the server software by uninstalling the existing version, and installing the required version of Rational® Test Automation Server, then you must perform the migration task after the installation of the server software is complete.</p> <p>For more information about the migration process, see Migrating user data on Ubuntu from 10.1.3 or later on page 88 or Migrating user data on Ubuntu from 10.1.2 or earlier on page 86.</p>
	10.1.2 or earlier		
Azure Kubernetes Service	10.2.3 or later	In-place upgrade	<p>When you upgrade the software by using the helm upgrade command, the migration of user data takes place during the upgrade process.</p>

Migrating user data on Red Hat OpenShift

When you want to use the data, which was in your earlier version, in a new version of IBM® Rational® Test Automation Server, you can migrate those user data.

Before you begin

You must have upgraded Rational® Test Automation Server to a new version.

About this task

You must replace {my-rtas} with the name of the release that you provided during the installation of the server software in all the instances in this procedure.



Remember: The `test-system` is the name of the namespace that you created during the installation of the server software. If you created a namespace by using a different value, then you must use that value in place of `test-system` in all the instances in this procedure.

1. Run the following script from the `ibm-rtas-prod/files` directory to create a directory that contains metadata related to the Persistent Volume Claims and their Persistent Volumes:

```
migrate.sh create-pvcs -n test-system {my-rtas}
```

2. Run the following script from the `ibm-rtas-prod/files` directory to merge the data into Rational® Test Automation Server:

```
migrate.sh merge-dbs -n test-system {my-rtas}
```

3. Run the following command to remove the resources that were created during the migration process:

```
migrate.sh delete-temp-resources -n test-system {my-rtas}
```

4. Run the following script from the `ibm-rtas-prod/files` directory to verify and test the installed server software:

```
ibm-rtas-prod/files/helm-test-diag.sh {my-rtas} -n test-system
```

Results

You have migrated the user data from the earlier version to the new version.

What to do next

You can perform the following tasks:

- Configure licenses for team spaces. See [Configuring licenses for team spaces on page 139](#).
- Configure the server software. See [Configuration of the server software on page 105](#).

Migrating user data on Ubuntu from 10.1.2 or earlier

When you want to use the data, which was in your earlier version, in a new version of IBM® Rational® Test Automation Server, you can migrate those user data.

Before you begin

You must have upgraded Rational® Test Automation Server to a new version.

About this task

The following table lists the variables that you must replace with the actual value in the command.

Variables	Description
{my-rtas}	The name of the release that you provided during the installation of the server software.

Variables	Description
{backup-release}	The Helm release name of the server where you store the backup data.
{target-release}	The Helm release name of the server where you want to restore the data.
{backup-file-name}	The name of the backed-up file that you saved.



Remember: The `test-system` is the name of the namespace that you created during the installation of the server software. If you created a namespace by using a different value, then you must use that value in place of `test-system` in all the instances in this procedure.

1. Run the following script from the `ibm-rtas-prod/files` directory to create a directory that contains metadata related to the Persistent Volume Claims and their Persistent Volumes:

```
migrate.sh create-pvcs -n test-system {my-rtas}
```

2. Run the following script from the `ibm-rtas-case/inventory/ibmRtasBase/files` directory to back up the data:

```
sudo backup.sh create-pvc-links -v ~/migration-pvc-links
```

3. Run the following command to stop the cluster and Rational® Test Automation Server:

```
k3s-killall.sh
```

4. Run the following script from the `ibm-rtas-case/inventory/ibmRtasBase/files` directory to restore the backed-up data:

```
sudo backup.sh restore -v ~/migration-pvc-links --release {backup-release}:{target-release}
{backup-file-name}
```

5. Run the following command to restart Kubernetes and to start Rational® Test Automation Server:

```
sudo systemctl start k3s
```

6. Run the following script from the `ibm-rtas-prod/files` directory to merge the data into the server:

```
migrate.sh merge-dbs -n test-system {my-rtas}
```

7. Run the following command to remove the resources that were created during the migration process:

```
migrate.sh delete-temp-resources -n test-system {my-rtas}
```

8. Run the following script from the `ibm-rtas-prod/files` directory to verify and test the installed server software:

```
ibm-rtas-prod/files/helm-test-diag.sh {my-rtas} -n test-system
```

Results

You have migrated the user data from the earlier version to the new version.

What to do next

You can perform the following tasks:

- Configure licenses for team spaces. See [Configuring licenses for team spaces on page 139](#).
- Configure the server software. See [Configuration of the server software on page 105](#).

Migrating user data on Ubuntu from 10.1.3 or later

When you want to use the data, which was in your earlier version, in a new version of IBM® Rational® Test Automation Server, you can migrate those user data..

Before you begin

You must have upgraded Rational® Test Automation Server to a new version.

About this task

The following table lists the variables that you must replace with the actual value in the command.

Variables	Description
{my-rtas}	The name of the release that you provided during the installation of the server software.
{backup-release}	The Helm release name of the server where you store the backup data.
{target-release}	The Helm release name of the server where you want to restore the data.
{backup-file-name}	The name of the backed-up file that you saved.

The `test-system` is the name of the namespace that is created during the setting up of the k3s environment.

1. Run the following script from the `ibm-rtas-case/inventory/ibmRtasBase/files` directory to create a directory that contains metadata related to the Persistent Volume Claims and their Persistent Volumes:

```
sudo backup.sh create-pvc-links
```

2. Stop the Kubernetes cluster by running the `k3s-killall.sh` command.
3. Restore the backed-up user data by running the following commands:

```
sudo ./backup.sh restore [options] {backup-file-name}
```

You can use the following parameters along with the restore command:

- `--namespace` or `-n`: Use this parameter to restore a specific namespace from the backup file. If you do not mention the namespace, then volumes from all the namespaces in the backup file are restored. You can map one namespace to another namespace by using a colon (:). The syntax is:

```
--namespace <name of the namespace> [:<target-namespace>]
```

- `--release` or `-r`: Use this parameter if the Helm release name of the server to which the backup is being restored is different than the Helm release name of the server where the backup was taken. The syntax is:

```
--release <backup-release>:<target-release>
```

- `--volumes` or `-v`: Use this parameter to specify the directory path of the volumes. The syntax is:

```
--volumes <path-of-the-directory>
```

- `-k` or `--confirm`: Use this parameter to skip the confirmation step.

4. Restart the Kubernetes cluster by running the `systemctl start k3s` command to start Rational® Test Automation Server.

5. Run the following script from the `ibm-rtas-prod/files` directory to create all the missing databases:

```
migrate.sh create-missing-dbs {my-rtas}
```



Note: You can expect the output of the command to contain several errors on objects that already exist. They do not indicate a problem with the execution of the script.

6. Run the following script from the `ibm-rtas-prod/files` directory to verify and test the installed server software:

```
ibm-rtas-prod/files/helm-test-diag.sh {my-rtas} -n test-system
```

Results

You have migrated the user data from the earlier version to the new version.

What to do next

You can perform the following tasks:

- Configure licenses for team spaces. See [Configuring licenses for team spaces on page 139](#).
- Configure the server software. See [Configuration of the server software on page 105](#).

Backup and restoration of the server data

You must back up all your server data before you uninstall the current version of the server software or upgrade to the latest version of the server software. You must back up the server data to avoid data loss or inaccessibility of data.

A backup is the process of creating a copy of the data on your system and storing it elsewhere, generally on secondary storage. You can then use that copy to recover if your original data is lost or becomes inaccessible.

A restore is a process of copying the backed-up data from the secondary storage and restoring it to the original location. You can restore the backed-up data when your original data is lost or becomes inaccessible.

You must back up and restore the data when you perform the following tasks:

- Move the existing environment to a new system.
- Change the name of the release or namespace that you used during the installation of the server software.
- Minimize the downtime of Rational® Test Automation Server during disaster recovery.
- Upgrade to the latest version of the server software.

To learn more about backup and restoration of the server data on platforms such as Red Hat OpenShift, Ubuntu, and Azure Kubernetes Service (AKS) refer to the following topics:

- [Backup and restoration of the server data on Ubuntu on page 93](#)
- [Backup and restoration of the server data on IBM Red Hat OpenShift on page 90](#)
- [Backup and restoration of the server data on Azure Kubernetes Service on page 97](#)

Backup and restoration of the server data on IBM® Red Hat OpenShift

To secure the data in Rational® Test Automation Server that is installed on the IBM® Red Hat OpenShift platform, you can back up the data. At any point in time, after you install Rational® Test Automation Server, you can restore the backed-up data.

Preparing IBM® Red Hat OpenShift cluster to backup and restore the server data

You must prepare your Red Hat OpenShift cluster before you back up or restore the data of Rational® Test Automation Server.

Before you begin

You must have installed Red Hat OpenShift CLI. For more information, refer to the Getting started with the CLI section in the [Red Hat OpenShift](#) documentation.

About this task

Velero is one of the tools that are available to back up and restore the data of Rational® Test Automation Server that is installed on the Red Hat OpenShift platform. You must prepare your cluster before you back up or restore the data by using Velero. You can also use the other tools to back up and restore the data. If you use a different tool, then you must include the Persistent Volumes in the cluster.

1. Log in to the cluster by using `oc login`.
2. Install and configure Velero with the Restic Integration.
3. Set the name of the namespace in which Rational® Test Automation Server is installed by running the following command:

```
NS=<namespace>
```

4. Update stateful sets to apply the annotations required by Velero to back up the PVs used by the pods by running the following commands:

```
for sts in $(oc get sts -n "$NS" -o name); do \
  if ! oc -n "$NS" patch --type=json "$sts" -p \
    '[{"op":"add","path":"/spec/template/metadata/annotations/backup.velero.io~1backup-volumes",
"value":"data"}]' 2>/dev/null; then
    oc -n "$NS" patch --type=json "$sts" -p \
      '[{"op":"add","path":"/spec/template/metadata/annotations",
"value":{"backup.velero.io/backup-volumes":"data"}}]'
  fi \
done
```

5. Change the security restriction of the pods by running the following commands:

```
for sts in $(oc get sts -n "$NS" -oname); do \
  if oc get -n "$NS" "$sts" -ojsonpath='{.spec.template.spec.securityContext}' | grep -q
"runAsNonRoot:true"; then \
    echo $sts
    oc patch -n "$NS" "$sts" --type json \
      -p '[{"op":"replace", "path":"/spec/template/spec/securityContext", "value":
{"runAsNonRoot": false}}]'
  fi \
done
```

Results

You have prepared your Red Hat OpenShift cluster to back up the server data.

What to do next

You can back up the server data. See [Backing up the server data on Red Hat OpenShift on page 91](#).

Related information

[Velero Documentation](#)

Backing up the server data on Red Hat OpenShift

When you want to upgrade to a new version of the server software or to move your existing environments to new systems, you must back up the server data.

Before you begin

You must have completed the following tasks:

- Been assigned the same role that was required to install and uninstall the server software.
- Installed the server software and all of the Rational® Test Automation Server pods are in `Running` state.
- Communicated to the users that Rational® Test Automation Server might be unavailable for some time until the process is complete.
- Prepared your cluster to back up and restore data by using Velero. See [Preparing IBM Red Hat OpenShift cluster to backup and restore the server data on page 90](#).

1. Log in to the cluster by using oc login.
2. Start the backup process by running the following command:

```
velero backup create <backup_file_name> --include-namespaces=<name_of_the_namespace>
```



Note: You must replace `<backup_file_name>` and `<name_of_the_namespace>` with the actual value in the command.

Results

You have backed up the data of Rational® Test Automation Server.

Related information

[Velero Documentation](#)

Restoring the server data on IBM® Red Hat OpenShift

You can restore the data that is backed up in Rational® Test Automation Server at any point in time after you install latest version of server software.

Before you begin

You must have completed the following tasks:

- Been assigned the same role that was required to install and uninstall the server software.
- Communicated to the users that Rational® Test Automation Server might be unavailable for some time until the process is complete.
- Backed up the data from Rational® Test Automation Server.

1. Log in to the cluster by using oc login.
2. Start the restore process by entering the following command:

```
velero restore create --from-backup=<backup-name> --restore-volumes
```

Results

You have restored the data of Rational® Test Automation Server.

Related information

[Velero Documentation](#)

Backup and restoration of the server data on Ubuntu

To secure the data of Rational® Test Automation Server that is installed on the Ubuntu server, you can back up the data. At any point in time, after you install Rational® Test Automation Server, you can restore the backed-up data.

Backing up the server data on Ubuntu

When you want to upgrade to a new version of the server software or to move your existing environments to new systems, you must back up the server data.

Before you begin

You must have completed the following tasks:

- Been assigned the same role that was required to install and uninstall the server software.
- Been granted with the `sudo` access.
- Installed the server software and all of the Rational® Test Automation Server pods are in `Running` state.
- Communicated to the users that Rational® Test Automation Server might be unavailable for some time until the process is complete.

1. Log in to the Ubuntu server and open a terminal.
2. Run the following command to retrieve the case from the IBM Cloud Pack repository:

```
curl -o ibm-rtas-case-10.1051.0.tgz
https://raw.githubusercontent.com/IBM/cloud-pak/master/repo/case/ibm-rtas-case/10.1051.0/ibm-rtas-case-10.1051.0.tgz
```

3. Run the following commands to fetch the scripts that are used to install Kubernetes:

```
tar xvf ibm-rtas-case-10.1051.0.tgz
chmod +x ibm-rtas-case/inventory/ibmRtasBase/files/*.sh
```

4. Change to the `ibm-rtas-case/inventory/ibmRtasBase/files` directory.

This directory contains the `backup.sh` script which is required to complete the backup operation.

5. Create a directory that contains metadata related to the Persistent Volume Claims and their Persistent Volumes by running the following command:

```
sudo ./backup.sh create-pvc-links
```

You can use the following optional parameter along with the `create-pvc-links` command:

`--volumes` or `-v`: Use this parameter to specify the directory path of the Volumes.

For example, `backup.sh create-pvc-links -v ibm-rtas-case/inventory/ibmRtasBase/files/my-pvc-links` creates a directory called `my-pvc-links` that stores the metadata required for backup.



Remember:



- If you did not provide the volume parameter, by default, the command creates a sub-directory in the same directory where you have the backup script.
- You must ensure that the directory you create must be empty or not exist to avoid script failure.
- If you specify a directory by using the `-v` parameter, then you must use the same parameter value when you restore the data.

6. Run the following command to stop the cluster and Rational® Test Automation Server:

```
k3s-killall.sh
```

7. Run the following command to create a backup of the existing user data:

```
sudo ./backup.sh create [options] <backup-file-name>
```

After you run this command, a backup of the local Persistent Volumes is created. The backup is created as tar archives that is compressed by using gzip (.tar.gz). The **create** command archives the Persistent Volumes into the *<backup-file-name>*.

You can use the following parameters along with the **create** command:

- `--namespace` or `-n`: Use this parameter to back up the Persistent Volumes in the specified namespace. If you do not mention the namespace, then all the Volumes from all the namespaces are included in the backup. The syntax is:

```
--namespace <name of the namespace>
```

- `--volumes` or `-v`: Use this parameter to specify the directory path of the Volumes. The syntax is:

```
--volumes <path-of-the-directory>
```



Remember:

- If you did not provide the volume parameter, by default, the command creates a sub-directory in the same directory where you have the backup script.
- If you specify a directory by using the `-v` parameter, then you must use the same parameter value when you restore the data.

For example,

```
sudo ./backup.sh create --namespace test-system my-backup.tar.gz
```

This command creates a backup file named *my-backup.tar.gz* that contains all of the Persistent Volumes associated with pods available in the *test-system* namespace.

8. Run the following command to restart the cluster and Rational® Test Automation Server:

```
sudo systemctl start k3s
```

Results

You have backed up the data of Rational® Test Automation Server.

Restoring the server data on Ubuntu

You can restore the data that is backed up in Rational® Test Automation Server at any point in time after you install latest version of server software.

Before you begin

You must have completed the following tasks:

- Been assigned the same role that was required to install and uninstall the server software.
- Been granted with the *sudo* access.
- Communicated to the users that Rational® Test Automation Server might be unavailable for some time until the process is complete.
- Backed up the data from Rational® Test Automation Server.

About this task

The following procedure is for the restoration of the server data from the current release. If you are upgrading the server software, then you can restore the server data from the earlier version to the latest version during the installation of the server software. For more information, see the **Related information** section.

1. Log in to the Ubuntu server and open a terminal.
2. Run the following command to retrieve the case from the IBM Cloud Pack repository:

```
curl -o ibm-rtas-case-10.1051.0.tgz
https://raw.githubusercontent.com/IBM/cloud-pak/master/repo/case/ibm-rtas-case/10.1051.0/ibm-rtas-case-10.1051.0.tgz
```

3. Run the following commands to fetch the scripts that are used to install Kubernetes:

```
tar xvf ibm-rtas-case-10.1051.0.tgz
chmod +x ibm-rtas-case/inventory/ibmRtasBase/files/*.sh
```

4. Change to the `ibm-rtas-case/inventory/ibmRtasBase/files` directory.
This directory contains the `backup.sh` script which is required to complete the restore operation.
5. Create a directory that contains metadata related to the Persistent Volume Claims and their Persistent Volumes by running the following command if you have not done before:

```
sudo ./backup.sh create-pvc-links
```

You can use the following optional parameter along with the **create-pvc-links** command:

`--volumes` or `-v`: Use this parameter to specify the directory path of the Volumes.

For example, `backup.sh create-pvc-links -v ibm-rtas-case/inventory/ibmRtasBase/files/my-pvc-links` creates a directory called `my-pvc-links` that stores the metadata required to restore your user data.



Remember:

- If you did not provide the volume parameter, by default, the command creates a sub-directory in the same directory where you have the backup script.
- You must ensure that the directory you create must be empty or not exist to avoid script failure.

6. Run the following command to stop the cluster and Rational® Test Automation Server:

```
k3s-killall.sh
```

7. Run the following command to restore the backed-up user data:

```
sudo ./backup.sh restore [options] <backup-file-name>
```

The **restore** command overwrites the existing Persistent Volumes with data from the *<backup-file-name>*.

You can use the following parameters along with the restore command:

- `--namespace` or `-n`: Use this parameter to restore a specific namespace from the backup file. If you do not mention the namespace, then volumes from all the namespaces in the backup file are restored.

You can map one namespace to another namespace by using a colon (:). The syntax is:

```
--namespace <name of the namespace> [ : <target-namespace> ]
```

- `--release` or `-r`: Use this parameter if the Helm release name of the server to which the backup is being restored is different than the Helm release name of the server where the backup was taken. The syntax is:

```
--release <backup-release> : <target-release>
```

- `--volumes` or `-v`: Use this parameter to specify the directory path of the volumes. The syntax is:

```
--volumes <path-of-the-directory>
```

- `-k` or `--confirm`: Use this parameter to skip the confirmation step.

For example,

```
sudo ./backup.sh restore --namespace test-system:new-test-system -r rel-1:rel-2 -k my-backup.tar.gz
```

This command restores the volumes that are backed up from the *rel-1* release in the *test-system* namespace to the *rel-2* release in the *new-test-system* namespace and skips the confirmation step.

8. Run the following command to restart the cluster and Rational® Test Automation Server:

```
sudo systemctl start k3s
```

**Note:**

- The cluster and Rational® Test Automation Server might take some time to restart and until that time, you cannot access the Rational® Test Automation Server URL.
- You can run the `kubectl get pods -A` command to verify that the Kubernetes environment is working. After a while, the status of the pods must be `Running` state.

Results

You have restored the data of Rational® Test Automation Server.

Related information

[Server software upgrade on Ubuntu on page 77](#)

Backup and restoration of the server data on Azure Kubernetes Service

To secure the data in Rational® Test Automation Server that is installed on Azure Kubernetes Service (AKS), you can back up the data. At any point in time, after you install Rational® Test Automation Server, you can restore the backed-up data.



Attention: The instructions provided are an example of how an AKS cluster can be configured to use Velero for backing up and restoring the Rational® Test Automation Server data. The procedures to backup and restore the server data might be not valid in all the scenarios.

You must refer to the Velero and Azure Kubernetes Service documentation to identify how you can configure Velero in your AKS cluster based on your existing AKS setup to backup and restore the Rational® Test Automation Server data.

- [Velero Documentation](#)

- [AKS Documentation](#)

Preparing the Azure Kubernetes Service cluster to back up the server data

When you want to back up the server data, you must prepare your Azure Kubernetes Service (AKS) cluster to store the backed-up data.

Before you begin

You must have completed the following tasks:

- Created an Azure subscription along with an **Owner** or **Contributor** and **User-Access Administrator** role on the Azure subscription.

You can create a subscription from [Microsoft Azure](#) if you do not have a valid subscription.

- Downloaded the Velero 1.5.3 plug-in for Microsoft Azure from the [Velero repository](#).



CAUTION: The instructions provided in this procedure are an example of how an AKS cluster can be configured to use Velero 1.5.3 Microsoft Azure plug-in for backing up and restoring the Rational® Test Automation Server data. The procedures to backup and restore the server data might be not valid in all the scenarios.

- Copied the Velero executable file into one of the directories in the **PATH** environment variable.
- Installed Azure Command-Line Interface (CLI). For more information, refer to the [Azure CLI](#) documentation.

About this task

Velero is one of the tools that is available to back up and restore the server data. You must prepare your AKS cluster where you installed Rational® Test Automation Server before you back up the data by using Velero.

1. Run the following command to log in to AKS interactively:

```
az login
```

Result

The command-line interface opens a browser and displays the Log-in page of AKS.

2. Sign in with your AKS account credentials.
3. Run the following command to set your active subscription:

```
az account set --subscription <subscription_name>
```

You must replace `<subscription_name>` with the name of your active Azure subscription.

4. Run the following command to set the location of your AKS cluster:

```
az configure --defaults location=<location_name>
```

For example, `az configure --defaults location=eastus2`



Tip: You can run the following command to get the list of all the available locations:

```
az account list-locations --query "[].{DisplayName:displayName, Name:name}" -o table
```

5. Run the following commands to create a storage account within the AKS cluster for the Velero to store the backup files:

```
AZURE_BACKUP_SUBSCRIPTION_NAME=<subscription_name>
AZURE_BACKUP_SUBSCRIPTION_ID=$(az account list
  --query="[?name=='$AZURE_BACKUP_SUBSCRIPTION_NAME'].id | [0]" -o tsv)
AZURE_BACKUP_RESOURCE_GROUP=Velero_Backups
```

```
az group create -n $AZURE_BACKUP_RESOURCE_GROUP --location <location_name>

AZURE_STORAGE_ACCOUNT_ID="velero$(date -u +%s)"
az storage account create \
--name $AZURE_STORAGE_ACCOUNT_ID \
--resource-group $AZURE_BACKUP_RESOURCE_GROUP \
--sku Standard_GRS \
--encryption-services blob \
--https-only true \
--kind BlobStorage \
--access-tier Hot
```



Note: You must replace `<subscription_name>` and `<location_name>` with the name of your active Azure subscription and location of the AKS cluster.

6. Run the following command to create a container within the storage account to store the backup files:

```
AZURE_BLOB_CONTAINER=velero
az storage container create -n $AZURE_BLOB_CONTAINER --public-access off --account-name
$AZURE_STORAGE_ACCOUNT_ID
```

7. Run the following commands to obtain a key of Azure storage account and to create a file that contains all the relevant environment variables:

```
AZURE_STORAGE_ACCOUNT_ACCESS_KEY=$(az storage account keys list \
--account-name $AZURE_STORAGE_ACCOUNT_ID \
--query "[?keyName == 'key1'].value" -o tsv)

cat << EOF > ./credentials-velero
AZURE_STORAGE_ACCOUNT_ACCESS_KEY=${AZURE_STORAGE_ACCOUNT_ACCESS_KEY}
AZURE_CLOUD_NAME=AzurePublicCloud
EOF
```



Note: There are several ways Velero can authenticate to Azure. This procedure is based on using a storage account access key. The available values for `AZURE_CLOUD_NAME` in the Storage account access key method are `AzurePublicCloud`, `AzureUSGovernmentCloud`, `AzureChinaCloud`, and `AzureGermanCloud`.

8. Run the following command to install Velero with the restic option by using the Azure storage account key:

```
velero install \
--provider azure \
--plugins velero/velero-plugin-for-microsoft-azure:v1.1.0 \
--bucket $AZURE_BLOB_CONTAINER \
--secret-file ./credentials-velero \
--backup-location-config
resourceGroup=$AZURE_BACKUP_RESOURCE_GROUP,storageAccount=$AZURE_STORAGE_ACCOUNT_ID,storageAccountKeyEnvVar=AZURE_STORAGE_ACCOUNT_ACCESS_KEY,subscriptionId=$AZURE_BACKUP_SUBSCRIPTION_ID \
--use-volume-snapshots=false \
--use-restic
```

Results

You have prepared your cluster to back up the server data.

What to do next

You can back up the server data. See [Backing up the server data on Azure Kubernetes Service on page 100](#)

Related information

[Velero Documentation](#)

[Velero plug-in for Microsoft Azure documentation](#)

[AKS Documentation](#)

Backing up the server data on Azure Kubernetes Service

You might want to back up the server data to protect against data corruption or malicious attacks. You might also want to back up the server data when you want to upgrade to a new version of the server software.

Before you begin

You must have completed the following tasks:

- Been assigned the same role that was required to install and uninstall the server software.
- Installed the server software and all of the Rational® Test Automation Server pods are in `Running` state.
- Communicated to the users that Rational® Test Automation Server might be unavailable for some time until the process is complete.
- Prepared your cluster to store the backed-up data by using Velero. See [Preparing the Azure Kubernetes Service cluster to back up the server data on page 97](#).

About this task

If you plan to back up the data immediately after you prepared your Azure Kubernetes Service (AKS) cluster, then you must go to [Step 3 on page 100](#) to start the backup process.

1. Run the following command to log in to AKS interactively:

```
az login
```

Result

The command-line interface opens a browser and displays the Log-in page of AKS.

2. Sign in with your AKS account credentials.
3. Run the following command to start the backup process of the Rational® Test Automation Server data:


```
velero backup create {backup_file_name} --include-namespaces=test-system
--default-volumes-to-restore
```



Note: You must replace `{backup_file_name}` with the name that you want to provide for the backup file.



Tip: You can run the `velero backup describe {backup_file_name} --details` command to verify the progress of the backup operation.

Results

You have backed up the data of Rational® Test Automation Server on AKS.

What to do next

You can restore the server data. See [Restoring the server data on Azure Kubernetes Service on page 101](#).

Related information

[Velero Documentation](#)

Restoring the server data on Azure Kubernetes Service

You can restore the server data that is backed up in Rational® Test Automation Server when required.

Before you begin

You must have completed the following tasks:

- Been assigned the same role that was required to install and uninstall the server software.
- Communicated to the users that Rational® Test Automation Server might be unavailable for some time until the process is complete.
- Backed up the data from Rational® Test Automation Server.

About this task

If you plan to restore the server data immediately after you backed up the data from Rational® Test Automation Server that you installed on Azure Kubernetes Service (AKS), then you must go to [Step 3 on page 101](#) to start the restoration process.

1. Run the following command to log in to AKS interactively:


```
az login
```


Result

The command-line interface opens a browser and displays the Log-in page of AKS.

2. Sign in with your AKS account credentials.
3. Run the following command to start the restore process of the Rational® Test Automation Server data:

```
velero restore create --from-backup={backup_file_name} --restore-volumes
```

 **Tip:** You can run the `velero restore describe {backup_file_name} --details` command to verify the progress of the restore operation.

 **Note:** You must replace `{backup_file_name}` with the name that you provided during the backing up of the server data.

Results

You have restored the data of Rational® Test Automation Server on AKS.

Related information

[Velero Documentation](#)

Uninstallation of the server software

When you no longer require Rational® Test Automation Server, you can uninstall the server software. You can uninstall the server software depending on the platform on which you installed the server software.

Uninstalling the server software from IBM® Red Hat OpenShift

When you want to install a new version of server software or you want to reinstall the server software when an ongoing installation fails, you can uninstall the server software and its components from the Red Hat OpenShift platform.

Before you begin

You must have completed the following tasks:

- Installed Rational® Test Automation Server software.
- Closed Rational® Test Automation Server, any open web browsers, and all other applications that are enabled by Rational® Test Automation Server.
- **Optional:** Backed-up data from the previous version of the product.

1. Open and log in to the terminal.
2. Run the following command to stop the workload that is running:

```
oc delete all,cm,secret -lexecution-marker -n test-system
```



Remember: The `test-system` is the name of the namespace. If you created a namespace by using a different value, then you must use that value in place of `test-system` in all the instances in this procedure.

3. Run the following command to uninstall the server software:

```
helm uninstall {my-rtas} -n test-system
```

The PersistentVolumeClaims and PersistentVolumes that were created during the installation are not deleted automatically. If you reinstall the server software, the user data is reused unless you specifically delete those volumes.

4. Run the following command to delete all user data that is contained in PersistentVolumeClaims and PersistentVolumes:

```
oc delete project test-system
```



Important: If you want to migrate data from a previous version to the latest version of the product, then you must ensure to back up the data before you run this command.

Results

You have uninstalled the server software from the Red Hat OpenShift platform.

Uninstalling the server software from Ubuntu

When you want to install a new version of server software or you want to reinstall the server software when an ongoing installation fails, you can uninstall the server software and its components from the Ubuntu platform.

Before you begin

You must have completed the following tasks:

- Installed Rational® Test Automation Server software.
- Closed Rational® Test Automation Server, any open web browsers, and all other applications that are enabled by Rational® Test Automation Server.
- **Optional:** Backed-up data from the previous version of the product.

About this task

You must replace `{my-rtas}` with the name of the release that you provided during the installation of the server software in all the instances in this procedure.

The `test-system` is the name of the namespace that is created during the setting up of the k3s environment.

1. Log in to the Ubuntu server using an SSH session.
2. Run the following command to stop the workload that is running:

```
kubectl delete all,cm,secret --lexecution-marker -n test-system
```

3. Run the following command to uninstall the server software:


```
helm uninstall {my-rtas} -n test-system
```

The PersistentVolumeClaims and PersistentVolumes that were created during the installation are not deleted automatically. If you reinstall the server software, the user data is reused unless you specifically delete those volumes.

4. Run the following commands to delete the entire Kubernetes environment, including all user data:

Ubuntu version	Commands to be run
18.04	<pre>cd ibm-rtas-case/inventory/ibmRtasBase/files sudo ./ubuntu-wipe.sh --confirm</pre>
20.04	<pre>cd ibm-rtas-case/inventory/ibmRtasBase/files sudo HOME=\$HOME ./ubuntu-wipe.sh --confirm</pre>

After you run this command, the system returns to the same state as if `ubuntu-init.sh` was never run.

 **Important:** If you want to migrate data from a previous version to the latest version of the product, then you must ensure to back up the data before you run this command.

Results

You have uninstalled the server software from the Ubuntu platform.

Uninstalling the server software from Azure Kubernetes Service

When you want to install a new version of server software or you want to reinstall the server software when an ongoing installation fails, you can uninstall the server software and its components from the Azure Kubernetes Service (AKS) cluster.

Before you begin

You must have completed the following tasks:

- Installed Rational® Test Automation Server software.
- Closed Rational® Test Automation Server, any open web browsers, and all other applications that are enabled by Rational® Test Automation Server.
- **Optional:** Backed-up data from the previous version of the product.

1. Run the following command to log in to AKS interactively:

```
az login
```

Result

The command-line interface opens a browser and displays the Log-in page of AKS.

2. Sign in with your AKS account credentials.
3. Run the following command to uninstall the server software:

```
helm uninstall {my-rtas} -n test-system
```

The PersistentVolumeClaims and PersistentVolumes that were created during the installation are not deleted automatically. If you reinstall the server software, the user data is reused unless you specifically delete those volumes.

4. Run the following command to delete the resource group from the AKS cluster:

```
az group delete -g <resource_group>
```

You must replace `<resource_group>` with the name of the resource group that you created during the creation of the AKS cluster.

Results

You have uninstalled the server software from the AKS cluster.

Configuration of the server software

You can find information about the tasks that you must perform after you installed the server software to configure Rational® Test Automation Server.

The following table describes the tasks that you can perform as an administrator after you installed the server software:

Requirement	Tasks	More information
Required	Review the default user management to know how the server software manages the users and their authentication.	User administration on page 106
Required	Import Certificate Authority (CA) into a browser to prevent certificate errors when you access the Rational® Test Automation Server UI.	Certificate authority: Importing and extending lists on page 115
Optional	Copy the third-party application Jars to Kubernetes to run API Suites that uses a transport.	Copying third-party application Jars to Kubernetes on page 118

Requirement	Tasks	More information
Optional	Change the password seed to provide enhanced security to Rational® Test Automation Server.	Changing the password seed on page 121
Optional	Disable the server extensions that are enabled during the installation of the server software if you do not want the server extensions to run on the cluster to save server resources.	Disabling server extensions on page 123

User administration

After you install IBM® Rational® Test Automation Server, you must consider how the software manages users and authentication.

You can review the default user management provided by the server and decide what additional controls you might want to add. If you manage users and authentication through an existing LDAP/AD server, you can review how to use that server to manage Rational® Test Automation Server users.

Enabling the Sign up link on the Login page of server

You can enable the **Sign up** link when it is not visible on the **Login** page of Rational® Test Automation Server by using the Keycloak UI.

Before you begin

You must have completed the following tasks:

- Installed Rational® Test Automation Server.
- Assigned the role of a Keycloak Administrator.
- Copied the URL of the Keycloak UI, username, and password on completion of the installation of server software.

About this task

The **Sign up** link on the **Login** page of Rational® Test Automation Server is not visible when you install the server software by using the default values. The **Sign up** link is hidden to provide the enhanced security to Rational® Test Automation Server. You can enable the **Sign up** link on the **Login** page when you want to provide self-registration of users in Rational® Test Automation Server.

You can enable the **Sign up** link on the **Login** page in two ways:

- You can use the `keycloak.signup.enabled` parameter during the installation of the server software.

For more information about installing the server software by using the `keycloak.signup.enabled` parameter, see the **Related information** section.

- You can use the Keycloak UI after the installation is complete.

For more information on Keycloak UI, see the **Related information** section.

The following procedure describes steps to be performed to enable the **Sign up** link by using Keycloak UI.

1. Log in to the Keycloak Admin Console.
2. Click **Realm Settings** from the left navigation pane.
3. Click the **Login** tab.
4. Set **User Registration** to **ON**.
5. Click **Save** to apply the changes.

Results

You have enabled the **Sign up** link on the **Login** page of Rational® Test Automation Server. The **Sign up** link is now visible on the **Login** page of Rational® Test Automation Server.

What to do next

You can click the **Sign up** link to create an account in Rational® Test Automation Server.

Related information

[Installing the server software on Azure Kubernetes Service on page 64](#)

[Installing the server software on IBM Red Hat OpenShift on page 33](#)

[Installing the server software on Ubuntu by using k3s on page 53](#)

[Keycloak](#)

Default user administration

You installed IBM® Rational® Test Automation Server and you want to know about how the software manages user access.

Rational® Test Automation Server uses Keycloak (<https://www.keycloak.org/>) to manage and authenticate users.

If you manage and authenticate users by using an LDAP and Active Directory server, you can configure Keycloak to connect to that server. For more information, see [LDAP user administration on page 109](#).

Keycloak uses the concept of a realm to manage and authenticate users. When you install the server software, a realm called `testserver` is created for you in Keycloak. All server users belong to this realm and when they log in to the server, they log into that realm.



Important: You must ensure that you have set the realm as `testserver` before you proceed with configuring the LDAP settings.

As an administrator, it is important to consider the following points about the server administration:

- By default, there is no administrator for Rational® Test Automation Server.

Such an administrator is required for accessing additional functions, which includes claiming ownership of server projects and unarchiving them. But you can assign administrative privileges to any user. You must do this by adding the admin role to the user in Keycloak.

- You must sign up a user that you want to be the administrator. You must go to the Login page at `https://<fully-qualified-dns-name>:443` and sign up.



Note: Do not use that admin user to perform non-administration tasks. Instead, sign up another user.

- After you sign up the user that you want to be the administrator for Rational® Test Automation Server, you must log in to the Keycloak Admin Console at `https://<fully-qualified-dns-name>:443auth/admin/master/console/#/testserver` to make that user the server administrator.

The default username for the Keycloak administrator is `keycloak`. The password is randomly generated when the software is installed. You can see the password by using the following `kubectl` command:

```
kubectl get secret -n test-system {my-rtas}-keycloak-postgresql -o jsonpath="{.data.password}" | base64 --decode; echo
```



Note: You must substitute `{my-rtas}` with the release name that you used during the installation of the server software.

After you log in to the Keycloak Admin Console, from the **Users** page, you can search and select the user that you want to make an administrator. From the **Groups** tab, you can join the user to the **Admins** group.

For more information about assigning user roles, see [Groups](#) in the Keycloak documentation.

Now that you are the server administrator, it is important to consider the following points about the default user management and authentication:

- Minimum password length defaults to 8 characters
- Email verification of new users is turned off
- The Forgot Password feature is turned on by default but no instructions are sent to the user to reset their password
- Forgotten user passwords are changed by you if you do not enable Keycloak to send instructions to reset a password

You can review the following sections about changing the default authentication controls.

Email settings

By default, the testserver realm sets the Forgot Password switch on. However, as an administrator, you must enable Keycloak to send an email to the user with instructions to reset their password. If you want to verify an email, you must also enable Keycloak to send an email to the user to verify their email address.

You must provide SMTP server settings for Keycloak to send an email. After you log in to the Keycloak Admin Console, see [Email Settings](#) in the Keycloak documentation.

Then, to set up the email verification, see [Forgot Password](#) in the Keycloak documentation.

Password policy

By default, the testserver realm has a password policy where the minimum length of a password is 8. As an administrator, you can update password policies in Keycloak.

After you log in to the Keycloak Admin Console, see [Password Policies](#) in the Keycloak documentation.

User password

If you did not enable Keycloak to send instructions to a user about how to reset a password, you must use the Keycloak Admin Console to change their password for them.

After you log in to the Keycloak Admin Console, see [User Credentials](#) in the Keycloak documentation.

User deletion

When a user is inactive or no longer needs to access the server, you can delete that user.

After you log in to the Keycloak Admin Console, see [Deleting Users](#) in the Keycloak documentation.

LDAP user administration

After you installed IBM® Rational® Test Automation Server, you can use a Lightweight Directory Access Protocol and Active Directory HTTP server to manage users and authentication. Some of the following best practices, can help you to use an LDAP/AD server to manage user access to Rational® Test Automation Server.

Rational® Test Automation Server uses Keycloak (<https://www.keycloak.org/>) to manage and authenticate users.

Existing user databases hold user credentials. Keycloak federates these existing external user databases through the concept of storage providers. By default, Keycloak supports an LDAP and Active Directory storage provider. By adding a storage provider, you can map LDAP user attributes into Keycloak. You can also configure more mappings.

Before you configure Keycloak to use an existing LDAP/AD provider, you might consider the following best practices:

- Set up your LDAP/AD provider as a read-only repository so that Rational® Test Automation Server cannot change it.
- Add and remove users in LDAP/AD and not the Keycloak local user database.

- Import and synchronize your LDAP/AD users to your Keycloak local database.
- Map a login style name, for example, `user1@server.com`, by using the `UserPrincipalName` attribute in LDAP/AD to a `username` in Keycloak. If you want the full name of the user as your login style, use the `cn` attribute in LDAP/AD.



Note: The LDAP/AD username attribute must match the LDAP/AD provider username attribute (**Username LDAP attribute**) in Keycloak for the LDAP/AD provider to connect with Keycloak.

The following sections use these best practices to guide you to set up Keycloak to connect to your LDAP/AD HTTP server.

LDAP provider selection in Keycloak

You can use the Keycloak Admin Console to add an LDAP/AD provider.

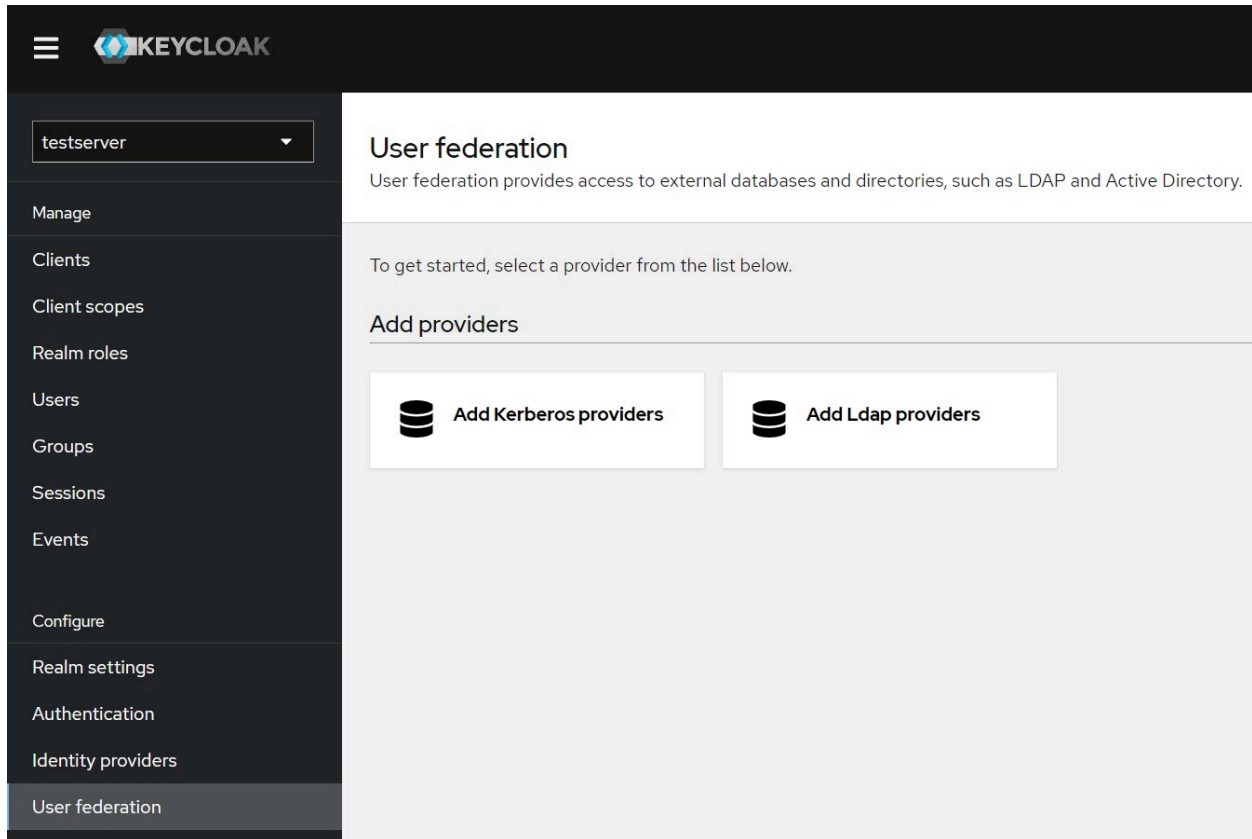
You can log in to the Keycloak Admin Console at `https://<fully-qualified-dns-name>auth/admin/master/console/#/testserver`. The default username for the Keycloak administrator is `keycloak`. The password is randomly generated when you installed the IBM® Rational® Test Automation Server software. You can see the password by using the following `kubectl` command:

```
kubectl get secret -n test-system {my-rtas}-keycloak-postgresql -o jsonpath="{.data.password}" | base64 --decode; echo
```



Note: You must substitute `{my-rtas}` with the release name that you used during the installation of the server software.

After you log in, you can go to the User Federation page to add your provider.



You must select the **Add Ldap providers** option and complete your LDAP/AD connection parameters.

Required settings for a successful connection to your LDAP/AD provider

In the **Add Ldap provider** page, you must complete the fields or select the required values from the list in the field to ensure a successful HTTP connection to your LDAP/AD provider.

Select or enter details in the fields that are displayed in the following sections on the **Add Ldap provider** page:

Section	Field	Action
General options	Console Display Name	Enter a name for the LDAP server that you can identify when it is displayed in the Keycloak Admin Console. For example, you can enter the name as LDAP.
	Vendor	Select the vendor that is providing the LDAP service from the list of providers. For example, you can select Active Directory from the list of vendors.
Connection and authentication settings	Connection URL	Enter the LDAP/AD URL so that Keycloak can connect to your LDAP/AD user database. For example, <code>ldap://<hostname>.<domain></code> .
	Enable StartTLS	Set to <i>On</i> , if you want to use a secure connection.

Section	Field	Action
	Test connection	Click to test the connection with the LDAP server.
	Bind type	Retain the default value set as <i>simple</i> .
	Bind DN	Enter the user ID of the LDAP/AD user database administrator. Make sure that you test the authentication and confirm that the authentication is successful.
	Bind credentials	Enter the password of the user ID of the LDAP/AD user database administrator.
	Test authentication	Click to test the authentication with the LDAP server database.
LDAP searching and updating	Edit mode	Select the <i>READ_ONLY</i> option from the list. This setting means that your LDAP/AD user database is read-only. No user data defined through the mapping of attributes in Keycloak such as the <code>username</code> is written back from the Keycloak user database to your LDAP/AD user database. This read-only setting also means that a new user cannot sign up from Rational® Test Automation Server as no user data can be written to your LDAP/AD user database.
	Users DN	Enter the directory where the LDAP users are listed.
	Username LDAP attribute	Retain the default value <code>cn</code> for the <code>username</code> , which is a first name, last name, or you can use <code>userPrincipalName</code> , which is <code>username@domain</code> . You might want a login that matches your company style. For example, you might prefer <code>joetester@mycompany.com</code> instead of Joe Tester. If you do make this change the <code>username</code> that you specify now must match the <code>username</code> in the Mapper. Keycloak makes this change for you.
	User object classes	Enter the User Object Classes as <i>User</i> because LDAP user records are found based on a user object class.
	User LDAP filter	Enter the filter so that a subset of all your LDAP users are imported as users by Keycloak.

Section	Field	Action
		For example, you can set up an RTAS user group for your LDAP/AD user database such that only those users are imported to Keycloak.

Mappers

Keycloak uses mappers to map the user attributes defined in the Keycloak user model such as **username** and **email** to the corresponding user attributes in the LDAP/AD user database. By default, when you saved your settings and created your LDAP/AD provider, the following mappers were created.

The username attribute that you specified in the **Username LDAP attribute** must match the **username** attribute defined in the Keycloak mapper for the LDAP/AD user database to connect with Keycloak.

Because you changed the **Username LDAP attribute** from the default value `cn` to `userPrincipalName`, Keycloak made the same change in the mapper called **username** to match.

User synchronization

You must import all users from your LDAP/AD user database by using the option to **Synchronize all users**. Users are imported based on your saved settings when you set up your LDAP/AD provider.

A successful import is followed by a success message with the number of users imported.

You can view all the LDAP/AD database users that were imported and authenticated from the **Users** page in the Keycloak Admin Console.

Users are listed with Username, Email, Last Name, and First Name. The value of the other attributes is fetched from the LDAP/AD user database by using mappers.



Important: If you get a message of a failed import for an LDAP/AD user, then the failed import might be due to any one of the following reasons:

- The **User logon name** field under the **Account** tab of that LDAP/AD user is blank.
- The value of the **Custom User LDAP Filter** field to import only a subset of all LDAP users does not match with the LDAP set of users.

You can check the log file of the Keycloak pod to understand the reason for the failed import.

You can perform the following steps to view the log file of the Keycloak pod:

1. Run the following command to view the list of all pods:

```
kubectl get pods -n test-system
```

2. Identify the Keycloak pod `ssocloak-0`.
3. Run the following command to view the log of the Keycloak pod:



```
kubectl logs {my-ots}-ssocloak-0 -n test-system
```



Note: Where, {my-ots} is the name of the release that is provided during the installation of the server software.

The log file of the Keycloak pod is displayed.

Managing account settings

You might want to change your password or other settings for your account.


About this task

As an account owner, you can view your account settings, which include your username, email, and first and last name. You can also change all of these settings except the username. Changing your password is also possible from your account settings.



Note: Account settings cannot be changed if IBM® Rational® Test Automation Server was configured to use an LDAP/AD provider. For more information about server security, see the related links.


- Edit your account details by following these steps:

1. Click the **User** icon  from the menu bar, and select **Account Settings**.

The **Edit account** page is displayed.

2. Edit your user details and save the changes. You can also reset your changes. If you cancel your changes, you return to the Home page.

- Change your password by following these steps:

1. Click the **User** icon  from the menu bar, and select **Account Settings**.

The **Edit account** page is displayed.

2. Click **Change password**.

The Change password page is displayed.

3. Type your current password followed by your new password

4. Confirm the new password by typing it again and save the changes. You can also reset your changes. If you cancel your changes, you return to the Home page.

Related information

[Default user administration on page 107](#)

Certificate authority: Importing and extending lists

When you want browsers and other applications to access Rational® Test Automation Server, you must enable browsers to trust the certificate authority (CA) of Rational® Test Automation Server. For some applications that integrate with Rational® Test Automation Server, you must also extend the CA lists that are trusted by the applications.

You can find the following tasks to import the certificate authority in the browser that you want to use:

- [Importing Certificate Authority into the Google Chrome browser on page 115](#)
- [Importing Certificate Authority into the Microsoft Edge browser on page 116](#)
- [Importing Certificate Authority into the Mozilla Firefox browser on page 116](#)

You can find the following tasks when you want to extend the CA lists:

- [Extending the trusted CA list on Linux on page 117](#)
- [Extending the trusted CA list on Windows systems on page 118](#)

Importing Certificate Authority into the Google Chrome browser

You must import certificate authority into the Google Chrome browser to prevent certificate errors when accessing the Rational® Test Automation Server UI.

Before you begin

You must have completed the following tasks:

- Saved a certificate authority (CA) during the installation of Rational® Test Automation Server.

You can run the following command to get the certificate from the system:

```
kubectl get secret ingress -n test-system -o jsonpath={.data.ca\\.crt} | base64 -d
```



Remember: *test-system* is the name of the namespace. If you created a namespace by using a different value, then you must use that value in place of the *test-system*.

- Installed the browser that you want to use to access Rational® Test Automation Server.
1. Open a Google Chrome browser.
 2. Navigate to **Customize and control Google Chrome > Settings**.
 3. Click **Security** from the **Privacy and security** pane.
 4. Click **Manage certificates**, and then **Import**.

5. Click **Next** in the **Certificate Import Wizard** window.
6. Click **Browse** and select the CA that you want to import, and then click **Next**.
7. Select the **Place all certificates in the following store** option to store the CA securely.
8. Click **Browse**, and then select **Trust Root Certification Authorities** as certificate store.
9. Click **OK**, and then **Finish**.

Results

You have imported the CA into the Google Chrome browser.

Importing Certificate Authority into the Microsoft Edge browser

You must import Certificate Authority (CA) into the Microsoft Edge browser to prevent certificate errors when accessing the Rational® Test Automation Server UI.

Before you begin

You must have completed the following tasks:

- Saved a certificate authority (CA) during the installation of Rational® Test Automation Server.

You can run the following command to get the certificate from the system:

```
kubectl get secret ingress -n test-system -o jsonpath={.data.ca\\.crt} | base64 -d
```



Remember: *test-system* is the name of the namespace. If you created a namespace by using a different value, then you must use that value in place of the *test-system*.

- Installed the browser that you want to use to access Rational® Test Automation Server.
1. Type `certmgr.msc` in the **Start** menu **Search** field, and then press **Enter**.
 2. Expand **Trusted Root Certification Authorities**, and then select **Certificate**.
 3. Right-click on the empty space, and then select **All tasks > Import**.
 4. Click **Next** in the **Certificate Import Wizard** window.
 5. Click **Browse** and select the CA that you want to import, and then click **Next**.
 6. Select the **Place all certificates in the following store** option to store the CA securely.
 7. Click **Browse**, and then select **Trust Root Certification Authorities** as certificate store.
 8. Click **Next**, and then **Finish** to import the certificate.

Results

You have imported the CA into the Microsoft Edge browser.

Importing Certificate Authority into the Mozilla Firefox browser

You must import Certificate Authority (CA) into the Mozilla Firefox browser to prevent certificate errors when accessing the Rational® Test Automation Server UI.

Before you begin

You must have completed the following tasks:

- Saved a certificate authority (CA) during the installation of Rational® Test Automation Server.

You can run the following command to get the certificate from the system:

```
kubectl get secret ingress -n test-system -o jsonpath={.data.ca\\.crt} | base64 -d
```



Remember: *test-system* is the name of the namespace. If you created a namespace by using a different value, then you must use that value in place of the *test-system*.

- Installed the browser that you want to use to access Rational® Test Automation Server.
 1. Open a Mozilla Firefox browser.
 2. Enter "about:preferences#privacy" in the address bar, and then press **Enter**.
 3. Locate and click the **View Certificates** option.
 4. Select **Authorities** tab, and then click **Import**.
 5. Select the CA that you want to import, and then click **Open**.
 6. Select **Trust this CA to identify websites** in the **Downloading Certificate** window.
 7. Click **OK** to import the CA into the browser, and then close the window.
 8. Restart Firefox.

Results

You have imported the CA into the Mozilla Firefox browser.

Extending the trusted CA list on Linux

Rational® Test Automation Server trusts the certificates signed by a recognized Certified Authority (CA). When you have any additional CAs, you must extend the trusted certificates list by using an environment variable on Linux.

Before you begin

You must have saved the CA assigned to Rational® Test Automation Server.



Note: You can contact the administrator of Rational® Test Automation Server if you do not have a copy of the CA.

1. Open the terminal.
2. Run the following command to create an environment variable on the Linux operating system:

```
export NODE_EXTRA_CA_CERTS= {path of the downloaded CA with the file extension}
```

For example,

```
export NODE_EXTRA_CA_CERTS=/opt/mycert/ca-master.crt
```

Results

You have extended the trusted CA certificates list on Linux.

Extending the trusted CA list on Windows systems


Rational® Test Automation Server trusts the certificates signed by a recognized Certified Authority (CA). When you have any additional CAs, you must extend the trusted certificates list by using an environment variable on Windows systems.

Before you begin

You must have saved the CA assigned to Rational® Test Automation Server.



Note: You can contact the administrator of Rational® Test Automation Server if you do not have a copy of the CA.

1. Right-click the Windows icon  from the Windows taskbar, and then click **System**.
2. Click **Advanced system settings** from the **Related setting** section.
3. Click the **Advanced** tab, and then click **Environment Variables**.
4. Click **New** from the **System variables** section.
5. Enter `NODE_EXTRA_CA_CERTS` in the **Variable name** field.
6. Enter the path of the downloaded CA with the file extension in the **Variable value** field.

For example, if you stored the `ca-master.crt` CA in the `C:\Users\Desktop\cert` location, then enter the **Variable value** field as `C:\Users\Desktop\cert\ca-master.crt`

7. Click **OK**.

Results

You have extended the trusted CA certificates list on Windows systems.

Copying third-party application Jars to Kubernetes

You can run API Suites in a project on Rational® Test Automation Server. If the API Suite uses a transport and the transport requires third-party application `Jar` files for a successful run, you must ensure that the third-party application `Jar` files are available at the test run time. To achieve this, you must copy the third-party application `Jar` files to the computer where Rational® Test Automation Server is installed on Kubernetes.

Before you begin

If you want to copy the third-party application `Jar` files to the remote Docker host, see [Copying third-party application Jars to a remote Docker host on page 223](#).

You must have server administrator or cluster administrator privileges.

You must have completed the following tasks:

- Identified the third-party application `jar` files that are required and copied the files. See [Test run considerations for API Suites or API tests on page 200](#).
- Copied the third-party application `jar` files from Rational® Integration Tester to the directory or folder in Kubernetes from where you can run the `kubectl` commands.

About this task

You can copy the third-party application `jar` files to the folder that is specific for the application under the `/data/<application_name>` folder. You need not extract the files.

You can perform this task any time after you have installed Rational® Test Automation Server and you plan to run an API Suite that meets any of the following conditions:

- The API Suite uses transports and the transports require third-party application `jar` files to be available at the test run time.
- The API Suite in the project has a results database configured.

You can get help on the `kubectl` commands by running the command: `kubectl cp --help` from the `/kube` directory. For more information, refer to the [kubectl documentation](#).

1. Use the following table to find the name of the folder that corresponds to the specific third-party application for the transport used in the API Suite.

Table 4. Name of the folder for the application

Application	Name of the folder to use
Camel	Camel
CentraSite	CentraSite
CICS	CICS
Coherence	Coherence
Database	JDBC
IMS	IMS
Integra	Integra
JMS	JMS
SAP RFC	SAP
Software AG Universal Messaging	SoftwareAGUM
TIBCO EMS	TIBCO
TIBCO Rendezvous	
TIBCO SmartSockets	

Application	Name of the folder to use
WebSphere Application Server Service Integration Bus (SiBus)	WAS
WebLogic	WebLogicJMX
Software AG webMethods	webMethods

- Open the command prompt from the `/kube` directory and copy the `Jar` files to the folder that corresponds to the application by using the following command:

```
kubectl cp <compressed_files> test-system/{my-rtas}-userlibs-0:/data/<application_name>/
```

For example, if you are copying the Database `Jar` files, then the name of the folder is `JDBC` and the command to use is:

```
kubectl cp mysql-connector-java.jar test-system/{my-rtas}-userlibs-0:/data/JDBC/
```



Note: You must substitute `{my-rtas}` with the release name that you used during the installation of the server software.

- Verify whether the `Jar` files are copied by running the following command:

```
kubectl exec {my-rtas}-userlibs-0 -n test-system -- ls /data/<folder_created>
```

For example, if you copied the Database `Jar` files, `mysql-connector-java.jar` to the folder `JDBC`, then the command to verify is:

```
kubectl exec {my-rtas}-userlibs-0 -n test-system -- ls /data/JDBC
```

The following information is displayed for the `JDBC` folder:

```
mysql-connector-java.jar
```



Note: You must substitute `{my-rtas}` with the release name that you used during the installation of the server software.

Results

You have successfully copied the third-party application `Jar` files to the folder in Kubernetes.

What to do next

You can configure the API Suite run. See [Configuring an API Suite or an API test run on page 267](#).

Changing the password seed

As an administrator, you can change the password seed that is used when you install the server software to provide enhanced security to Rational® Test Automation Server.

Before you begin

You must have completed the following tasks:

- Installed Rational® Test Automation Server. See [Installation of the server software on page 29](#).
- Installed the JSON command-line tool, jq, and ensured that the jq is in your environment PATH. For information about jq, refer to [jq documentation](#).
- Installed the Curl command line tool. For more information refer to [curl documentation](#).


About this task

When you install Rational® Test Automation Server, you supply a password seed when you run the `helm install` command. This password seed is used to generate several Kubernetes secrets. Kubernetes Secrets can contain the following information:

- The authentication credentials for micro-services.
- An encryption key for the user-created secrets collection or other secrets.

When you change the password seed for Rational® Test Automation Server, you must consider the following scenarios:

- Rational® Test Automation Server cannot communicate until you reconcile the passwords which are in Kubernetes Secrets by using the old and a new password seed.
- Users cannot read secret collections or other secrets that they have created in Rational® Test Automation Server until you re-encrypt them using a new password seed.

 **Important:** You must provide an offline token and old password seed that you used during the installation of server software to re-encrypt user secrets.

1. Run the following command to change the password seed for Rational® Test Automation Server:

```
helm upgrade {my-rtas} ./ibm-rtas-prod -n test-system \
--reuse-values \
--set global.ibmRtasPasswordAutoGenSeed={my-new-super-secret}
```



Notes: You must substitute the value of the following variables with the actual value in the command:



- `{my-rtas}` with the release name that you used during the installation of the server software.
- `{my-new-super-secret}` with a new value of your choice as the password seed.
- You must run the following `helm upgrade` command from the same directory where the `helm install` command was run during the installation of the server software. Because the upgrade is dependent on the helm charts and `.yaml` file values used during the run time of the `helm install` command.

2. Run the following script to generate new server secrets from the updated password seed and to save them to the persistent storage:

```
./ibm-rtas-prod/files/reconcile-secrets.sh -n test-system {my-rtas}
```

3. Run the following command to restart all the pods:

```
kubectl delete pods -n test-system \
-lapp.kubernetes.io/instance={my-rtas} \
-lapp.kubernetes.io/managed-by=Helm
```

4. Run the following commands to re-encrypt the user-created secrets collection or other secrets by providing the old password seed:

```
export ACCESS_TOKEN=$(curl -k -X POST {SERVER_URL}/rest/tokens/ \
-H "Content-Type: application/x-www-form-urlencoded" \
-H "accept: application/json" \
-d "refresh_token={OFFLINE_TOKEN}" | jq -r '.access_token')

curl -k -X POST {SERVER_URL}/rest/secrets/re-encrypt/ \
-H "Authorization: Bearer $ACCESS_TOKEN" \
-H "Content-Type: application/json" \
-d "{\"type\":\"helm\",\"password_auto_gen_seed\":\"{OLD_SEED}\"}"
```



Note: You must substitute the value of the following variables with the actual value in the following commands:

- `{SERVER_URL}` with the URL of your Rational® Test Automation Server UI.
- `{OFFLINE_TOKEN}` with the offline token that belongs to a user with the administrator role.
- `{OLD_SEED}` with the previous password seed that you used during the installation of the server software.

5. Run the following command to display the log file of the gateway pod:

```
kubectl logs {my-rtas}-gateway-abcdefghij-abcde -n test-system
```



Note: You must substitute the value of the following variables with the actual value in the command:



- `{my-rtas}` with the release name that you used during the installation of the server software.
- `abcdefghijkl-abcde` with an identifier that is assigned to the name of the gateway pod.

You can run the `kubectl get pods -n test-system` command to obtain the identifier that is assigned to the gateway pod.

Result

The following message is displayed when re-encryption is completed:

```
reEncrypt complete. StringyReEncryptor [total=100, fixed=100, broken=0, noop=0]
```

Results

You have successfully changed the password seed for Rational® Test Automation Server.

Server extensions

You can run specific tests in Rational® Test Automation Server by enabling the server extensions.

The server extensions are enabled at the time of installation of the server software. When the server extensions are enabled, you can perform the following tasks on Rational® Test Automation Server:

- Add repositories that contain the test assets for the HCL AppScan CodeSweep tests, JMeter tests, JUnit tests or Postman collections.
- Configure a test run for the following test types:
 - **HCL AppScan CodeSweep**
 - **JMeter**
 - **JUnit**
 - **Postman**

To learn more about enabling and disabling the server extensions, refer to the following topics:

Disabling server extensions

When you do not want the server extensions to run on the cluster to save server resources, you can disable server extensions.

Before you begin

You must have ensured that the server extension that you want to disable is enabled and running.



Tip: You can verify the status of the extension by running the `kubectl get pods -n {name_of_the_namespace}` command. The status of an enabled server extension is displayed as `Running`.

About this task

You must substitute `{my-rtas}` with the release name that you provided during the installation of the server software in the following procedure.

The `test-system` is the name of the namespace. If you created a namespace by using a different value, then you must use that value in place of `test-system` in all the instances in this procedure.

1. Log in to the platform that you used to install the server software.

For example, you can log in to the OpenShift Container Platform (OCP) cluster if you installed the server software on Red Hat OpenShift.

2. Run the following command to disable all the extensions:

```
helm upgrade {my-rtas} -n test-system ./ibm-rtas-prod \
--reuse-values \
--set global.ibmRtasJmeterEnabled=false \
--set global.ibmRtasJUnitEnabled=false \
--set global.ibmRtasCodesweepEnabled=false \
--set global.ibmRtasPostmanEnabled=false
```

If you want to disable any specific extension, then you can set the respective helm parameter value to `false`.

For example, to disable the JMeter extension alone, then you can run the following command:

```
helm upgrade {my-rtas} -n test-system ./ibm-rtas-prod \
--reuse-values \
--set global.ibmRtasJmeterEnabled=false
```

3. Run the following command to verify the status of the extensions:

```
kubectl get pods -n test-system
```



Note: The pod for each server extension that you disabled no longer displays.

Results

You have disabled the required server extensions on Rational® Test Automation Server.

Enabling server extensions

If you find that the server extension is disabled and you want to run the tests supported by the server extension, then you must enable them.

Before you begin

You must have ensured that the server extension that you want to enable is disabled.

About this task

You must substitute `{my-rtas}` with the release name that you provided during the installation of the server software in the following procedure.

The `test-system` is the name of the namespace. If you created a namespace by using a different value, then you must use that value in place of `test-system` in all the instances in this procedure.

1. Log in to the platform that you used to install the server software.

For example, you can log in to the OpenShift Container Platform (OCP) cluster if you installed the server software on Red Hat OpenShift.

2. Run the following command to enable all the extensions:

```
helm upgrade {my-rtas} -n test-system ./ibm-rtas-prod \
--reuse-values \
--set global.ibmRtasJmeterEnabled=true \
--set global.ibmRtasJUnitEnabled=true \
--set global.ibmRtasCodesweepEnabled=true \
--set global.ibmRtasPostmanEnabled=true
```

If you want to enable any specific extension, then you can set the respective helm parameter value to `true`.

For example, to enable the Postman extension alone, then you can run the following command:

```
helm upgrade {my-rtas} -n test-system ./ibm-rtas-prod \
--reuse-values \
--set global.ibmRtasPostmanEnabled=true
```

3. Run the following command to verify that specific or all the extensions are enabled:

```
kubectl get pods -n test-system
```

Result

The status of the specific extension or all the extensions in the pod is displayed as `Running`.

Results

You have enabled the required server extensions on Rational® Test Automation Server.

Related information

[Configuring a run of a JMeter test on page 344](#)

[Configuring a run of a JUnit test on page 353](#)

[Configuring a run of a Postman test on page 362](#)

[Configuration of a scan of an application code on page 383](#)

Team space administration

You can find information about team spaces including the initial team space on IBM® Rational® Test Automation Server along with the tasks that you can perform to administer a team space.

You can find the following information about team spaces:

- [Team space overview on page 126](#)
- [Creating a team space on page 129](#)
- [Tasks in a team space on page 131](#)
- [Tasks in projects in a team space on page 134](#)
- [Management of licenses in Rational Test Automation Server on page 136](#)

Team space overview

Team spaces are logical partitions of IBM® Rational® Test Automation Server. Team space is an exclusive, separate, and secure space on Rational® Test Automation Server that enables you to share a single installation among multiple teams.

You can find the following information about team spaces:

- [Initial team space on page 126](#)
- [Team spaces on page 127](#)
- [Migration from earlier releases of Rational Test Automation Server on page 128](#)

Initial team space

After you install Rational® Test Automation Server and log in for the first time, you enter the **Initial team Space**.

The initial team space is the default logical partition that is created on an instance of Rational® Test Automation Server.

As a server administrator, you must consider the following points:

- After you install Rational® Test Automation Server and log in for the first time, the **Projects** page of the initial team space is displayed.
- You must configure the license server for the initial team space.
- Add or remove users in the team space and assign them different roles in a team space.
- You can assign the role of a *Team Space Owner* to any other member in the initial team space.
- Change the type of the initial team space from the default type of *permissive*.

The following table describes the type that you can assign to a team space:

Type	Description
Permissive	Specifies that any licensed user of Rational® Test Automation Server can access a team space and get assigned the default roles of a <i>Project Owner</i> and <i>Member</i> in the team space.
Restrictive	Specifies that all licensed users of Rational® Test Automation Server must request for access to a team space and must be provided access and assigned roles by the <i>Team Space Owner</i> .

Type	Description
	The Restrictive type of team space can be either Public or Private . A Public team space is visible in the team spaces dashboard, while a Private team space is not visible in the team spaces dashboard.

- You can evaluate the requirement and create team spaces other than the initial team space.

As a licensed user, you must consider the following points:

- When you log in for the first time immediately after the installation of Rational® Test Automation Server, the **Projects** page of the initial team space is displayed.
- After you log in to the initial team space, you are assigned the default roles of *Project Creator* and *Member* of the initial team space.
- On subsequent logins, the **My Projects** page in the initial team space is displayed as the default page.

Team spaces

You can either work with the initial team space or you can create multiple team spaces on Rational® Test Automation Server. The team space facilitates users of the same team space to work together.

Apart from being a container for projects, a team space facilitates the registration of remote Docker hosts and Resource Monitoring agents, designing the system model, and viewing the registered agents, Dockers, or intercepts.

After you create a team space as a **server administrator**, you must consider the following actions:



Note: You are assigned the default role of a *Team Space Owner* for team spaces that you create.

- Add users in the team space and assign them different roles in a team space.
- Assign the role of a *Team Space Owner* to other team space members, if you want other members to own the team space.



Note: As a *Team Space Owner*, you can remove any member of your team space from the assigned role. The member who is removed from a team space, can send a request and become a member of that team space after the request is accepted.

- If the *Team Space Owner* is no longer a member of a team space, then you as the server administrator can access that team space with the default role of a *Team Space Owner*. You can then assign the role of a *Team Space Owner* to any other member.

- Delete the initial team space or any other team space that you created, if required.



Note: If you delete the initial team space, the users of the initial team space are redirected to the **Team Space Dashboard** page.

You as a server administrator or the *Team Space Owner* must perform the following tasks in a team space before other members can start using the team space:

- Configure the license server.
- Configure the team space type as either *Restrictive* or *Permissive*.
- Configure the SMTP server and add members who can receive notifications about the operations performed in the team space.

As a member of a team space, you must consider the following points:

- When you log in for the first time immediately after the installation of Rational® Test Automation Server, the **Projects** page of the team space is displayed.
- After you log in to the team space, you are assigned the default roles of a *Project Creator*, and *Member* of the team space.
- You can request to become a member of any other team space. After your request is approved, you are assigned the default role of a *Member* of that team space.
- You can become a member of either a **Permissive** or **Restrictive** type of team space.
- On subsequent logins, the **Team Space Dashboard** page is displayed as the default page.

Migration from earlier releases of Rational® Test Automation Server

The initial team space is the only logical partition in Rational® Test Automation Server 10.1.3. If you migrate from an earlier release of Rational® Test Automation Server to 10.2 or later, then all the assets and resources that you created and associated with the initial team space are migrated to associate with the initial team space in Rational® Test Automation Server 10.2 or later.



Note:

The results of migration are as follows:

- All projects are contained in the initial team space.
- If you created a repository in a team space, then you can find the repository in the initial team space in 10.2.
- All Dockers, agents, and intercepts that you registered with the server are then registered with the initial team space and you can view from the **Agents and Intercepts** page of the initial team space.
- If you configured a resource monitoring agent in any project, then the details of the configured agent are available in the project.
- You can find the details of SMTP configuration on the server. Only you as a server administrator can view the details of SMTP configuration.

- The license server that is configured for the server for earlier release of Rational® Test Automation Server is then configured for the initial team space.
- All users are assigned the role of a *Project Creator* and *Architect* of the initial team space.
- All users with server administrator roles are assigned the default role of a *Team Space Owner* of the initial team space.



Notes:

- All users of earlier releases are migrated to Rational® Test Automation Server 10.2 or later. You can view the list of all migrated users in the application only when they log into the application at least once.
- After you migrate the server data from a previous version to a newer version, the **Project Owner** must log in to Rational® Test Automation Server so that other members of the project can access the test assets in that project.

Related information

[Team space administration on page 125](#)

[Default user administration on page 107](#)

Creating a team space

To manage projects in the logical containers of IBM® Rational® Test Automation Server, you must create team spaces.

Before you begin

You must have been assigned the role of a server administrator of Rational® Test Automation Server.


About this task

As a server administrator, you can create multiple team spaces on Rational® Test Automation Server.

1. Log in to Rational® Test Automation Server.

Result

The **Projects** page of the initial team space is displayed.

2. Click the **Settings** icon  from the left navigation pane, and then click **Create team space**.

Result

The **New Team Space** dialog is displayed.

3. Provide a name to the team space.

Result

The **URL alias** field is automatically filled.



Note: If you modify the name of a team space, the **URL alias** is also modified automatically.

4. **Optional:** Modify the **URL alias** for the team space, if required.



Note: You must enter the **URL alias** that starts with a letter.

5. Select an option to set the permissions to access the team space. You can select any one of the following options:

- **Permissive** – When a team space is set as **Permissive**, the new user gets the permissions automatically on sending the request to become a member of that team space.
- **Restrictive** – When a team space is set as **Restrictive**, the *Team Space Owner* provides permission to the new user who becomes a member of that team space.

You can set the visibility mode of the **Restrictive** type team space as **Public** or **Private**.

- The **Public** team space is visible to all users in the **Team Space Dashboard**.
- The **Private** team space is visible only to the members of that team space in the **Team Space Dashboard**.



Note: If you select the **Restrictive** type team space, then the default setting of the team space becomes **Public**.

6. Provide a brief description of the team space.

7. Select the owner of the team space from the list of users who accessed the application.

The selected user is assigned the role of a *Team Space Owner*.



Note: You can assign the *Team Space Owner* role to only one user when you create a team space.

8. Click **Create**.

Result

The dashboard of the team space is displayed.

Results

You have created a team space.

If you are assigned the role of a *Team Space Owner*, then you can view the name of the team space that you created in the navigation pane.

If you assigned any other licensed user with the role of a *Team Space Owner*, then you can view the team space under the **Other Team Spaces** panel.

What to do next

You must configure the license for your team space. See [Configuring licenses for team spaces on page 139](#) .

Tasks in a team space

You can find information about the tasks that you can perform as a *Member*, *Project Owner*, *Architect*, or *Team Space Owner* in a team space on IBM® Rational® Test Automation Server.

The roles assigned in a team space for users are as follows:

Persona	Initial team space		Any team space	
	Role	Notes	Role	Notes
Server administrator	<i>Team Space Owner</i>	Default role assigned	<i>Team Space Owner</i>	Default role assigned
Licensed user	<i>Team Space Owner</i>	Role assigned by Server administrator	<i>Team Space Owner</i>	Role assigned by Server administrator
	<i>Architect</i>	Role assigned by <i>Team Space Owner</i>	<i>Architect</i>	Role assigned by <i>Team Space Owner</i>
	<i>Project Creator</i>	Default role assigned	<i>Project Creator</i>	<ul style="list-style-type: none"> • Default role assigned in a <i>Permissive</i> team space. • Role assigned by <i>Team Space Owner</i> in a <i>Restrictive</i> team space.
	<i>Member</i>	Default role assigned	<i>Member</i>	Default role assigned in either a <i>Permissive</i> or <i>Restrictive</i> team space.


The following table shows the tasks that can be performed by licensed users with their assigned roles in a team space:

Tasks	Team Space Owner	Architect	Project Creator	Member
Configuring licenses	✓			
Configuring a Simple Mail Transfer Protocol (SMTP) server to send email notifications.	✓			
Changing the license server configuration	✓			
Adding members in a team space	✓			

Tasks	Team Space Owner	Architect	Project Creator	Member
Assigning different roles to other team space members	✓			
Modifying configuration details of a team space	✓			
Deleting a team space	✓			
Renaming a team space	✓			
Adding or removing the team space repository from a team space	✓	✓		
Designing a system model	✓	✓		
Creating projects in a team space	✓		✓	
Adding, removing, or modifying remote Docker hosts	✓	✓	✓	✓
Adding agents	✓	✓	✓	✓
Adding, modifying, or removing resource monitoring sources	✓			
Connecting resource monitoring agents	✓			
Adding resource monitoring labels	✓			
Working with projects in a team space	✓	✓	✓	✓

The following table provides the links to the information about the tasks that can be performed by users with different roles in a team space:

Tasks	Roles	More information
Configuring a team space license server	<i>Team Space Owner</i>	Configuring licenses for team spaces on page 139
Configuring the Prioritized Floating Users	<i>Team Space Owner</i>	Adding users to the Prioritized Floating Users list on page 141
Modifying the configuration of a team space	<i>Team Space Owner</i>	Modifying the configuration of a team space on page 607
Viewing the configuration of a team space	All members of a team space	Viewing the configuration of a team space on page 609

Tasks	Roles	More information
Reassigning the role of team space members	<i>Team Space Owner</i>	Managing members and their roles in a team space on page 612
Becoming a team space member	<ul style="list-style-type: none"> • <i>Team Space Owner</i> • All users can send a request 	Becoming a team space member on page 611
Viewing team spaces	<p>All members of a team space</p> <p> Notes:</p> <ul style="list-style-type: none"> • The Permissive and public Restrictive team spaces are visible to all the users. • The private Restrictive team spaces are only visible to its members. 	Viewing team spaces on page 603
Deleting a team space	<i>Team Space Owner</i>	Deleting a team space on page 615
Adding a project in a team space	<i>Project Creator</i>	Adding a project on page 619
Viewing existing projects	All members of a team space	Viewing active projects on page 624
Viewing details about the users or license server	All members of a team space	Viewing license details of the team space on page 143
Installing remote agents and configuring them to register with Rational® Test Automation Server	All members of a team space	Management of performance agents on page 210
Registering a remote Docker host with Rational® Test Automation Server	All members of a team space	Registering a remote Docker host on page 225
Editing configurations of a remote Docker host	All members of a team space	Editing configurations of a remote Docker host on page 229
Unregistering a remote Docker host	All members of a team space	Unregistering a remote Docker host from IBM Rational Test Automation Server on page 233

Tasks	Roles	More information
Viewing remote agents, remote Dockers, or intercepts that are registered with Rational® Test Automation Server in the initial team space	All members of a team space	<ul style="list-style-type: none"> • Viewing remote Docker hosts that are registered with IBM Rational Test Automation Server on page 227 • Viewing performance agents that are registered with Rational Test Automation Server on page 212 • Viewing intercepts that are registered with a team space on IBM Rational Test Automation Server on page 415
Adding resource monitoring sources and agents	<i>Team Space Owner</i>	Resource monitoring on page 487
Adding resource monitoring labels to the sources	<i>Team Space Owner</i>	Controlling resource monitoring sources in a schedule on page 521
Viewing the performance metrics	All members of a team space	Viewing resource monitoring usage data on page 528
Creating a model of the system under test	<i>Architect</i>	Creating a system model on page 175
Adding a repository to a team space	<ul style="list-style-type: none"> • <i>Team Space Owner</i> • <i>Architect</i> 	Adding a repository to a team space on page 173
Viewing an existing system model	All members of a team space	Viewing the system model on page 197

Related information

[Team space administration on page 125](#)

Tasks in projects in a team space


You can find information about the roles that you can assign in a project for members of a team space and the tasks that you can perform in projects in a team space.

Members of a team space with the role of a *Project Creator* can create projects in the team space and become the *Owner* of a project. The owners of projects can add *Members* of the team space to a project and assign them the following roles:

- *Owner*
- *Tester*
- *Viewer*

For the list of actions that members assigned any of the roles can perform, see [Managing access to server projects on page 627](#).

All projects are created in a team space. When you open a project, the options that are displayed in the navigation pane are specific to the projects within a team space.

When you are working on a project in a team space and you want to return to the options available to work in the team space, you can click the **Home** icon  in the navigation pane.



Note: You must be assigned the role of an *Owner*, a *Tester*, or a *Viewer* of a project.

The following table lists the tasks that members of a team space can perform while working with projects in a team space:

Tasks	More information...
Viewing projects in a team space.	Viewing active projects on page 624
Adding a project.	Adding a project on page 619
Adding a repository that contains test assets and resources that you want to run on the server.	Adding repositories to a server project on page 620
Adding secrets to a secrets collection.	Secrets configuration on page 623
Encrypting the data set resources.	Data security
Configuring a change management system.	Configuration of a change management system on page 532
Adding members to a project and configuring their roles.	Adding users to a project on page 623
Selecting the global branch in a project to view the test assets and resources.	Selecting the global branch in a project on page 638
Adding agents to your project, if you are using remote agents as a location to run certain tests.	Adding a performance agent to a project on page 213
Adding remote Dockers, if you are using a remote Docker host as a location to run tests.	Adding a remote Docker host to the project for running tests on page 228

Tasks	More information...
Reading the considerations that you must take into account before you configure a test run.	Prerequisites to running tests on page 198
Reading the considerations that you must take into account before you configure a run or start a virtual service.	Prerequisites for running virtual services on page 410
Configuring a run of the test assets.	Test run configurations on page 249
Configuring a run of the virtual service resource.	Configuring a run of a virtual service on page 419
Viewing the progress of a test run.	Viewing the progress of running test assets on page 398
Managing a running test.	Management of running tests on page 394
Viewing the results of a test run after the run is completed.	Test results on page 455
Reviewing the test results and creating defects for test runs from the Results page.	Creating Jira issues for tests that ran on page 538

Management of licenses in Rational® Test Automation Server

As a *Team space Owner* or *Server Administrator*, you can configure and manage licenses for your team spaces in Rational® Test Automation Server. As a member of any team space, you can only view the number of licenses that are configured for the team space. If you are not a member of any team space, you cannot view or perform any operations in any of the team spaces on Rational® Test Automation Server.

You can get all the information of configured license server for different team spaces from various *Team Space Owners*. You can navigate to the Rational® Test Automation Server Swagger UI by accessing the following URL:

`https://myorg.com/#/Initial/admin/docs`

Where `myorg.com` is the hostname of Rational® Test Automation Server.

You can then open the **Licensing management** tab, and then click the **GET /rest/licensing/inventory** option. To generate the report that contains all the information of license server, you can click the **Try it out** option, and then click **Execute**.

The license inventory report contains information such as details about the configuration of team space licensing, a list of *Team Space Owners* and their email addresses, information about licenses that are mapped to the license server, and the current connectivity status of the license server.

Depending on your privileges, you can perform the following tasks in Rational® Test Automation Server:

Privileges	Tasks	More information
<i>Team space Owner</i> OR <i>Server Administrator</i>	Read about licensing on Rational® Test Automation Server.	Server software licensing information on page 137
	Configure licenses.	Configuring licenses for team spaces on page 139
	Configure prioritized users.	Adding users to the Prioritized Floating Users list on page 141
	Remove or replace prioritized users.	Removing or replacing prioritized users on page 142
Licensed user	Perform tasks or operations in a team space.	Tasks in a team space on page 131
	Viewing details about licenses.	Viewing license details of the team space on page 143

Server software licensing information

You can find information about the licensing on Rational® Test Automation Server.

To use the features offered by Rational® Test Automation Server, you must purchase licenses. You can then install the licenses on Rational License Key Server and configure the details of Rational License Key Server on the **License Configuration** page of Rational® Test Automation Server.

Rational® Test Automation Server requires an additional license when you use the service virtualization feature. You must have sufficient **Rational® Test Virtualization Server Concurrent Virtual Service** licenses to run instances of virtual services that are contained in API Suites on Rational Test Automation Server.

A single **Concurrent Virtual Service** license is used from the total number of **Rational® Test Virtualization Server Concurrent Virtual Service** licenses that are configured on Rational License Key Server for Rational® Test Virtualization Server when any of the following events occur:

- An instance of a virtual service is started by running a virtual service resource. A license is consumed for every running instance of the virtual service.
- An API suite is run that has a scenario that references a local virtual service and an instance of that virtual service is run.

The **Concurrent Virtual Service** license is returned to the license pool when the virtual service instance stops.

As a *Team space Owner*, you can add users to the **Prioritized Floating Users** list from the **License Configuration** page. The prioritized floating users do not return their floating licenses to the license pool when they log out from Rational® Test Automation Server. Rational License Key Server ensures that the licenses are always available for the prioritized

users. This feature is useful for those users who generate offline tokens that are used for long-run tests even when the members are logged out of Rational® Test Automation Server.



Remember: A single floating license is used from the total number of floating licenses when you perform most of the operations in Rational® Test Automation Server.



Important: You can perform only the following operations in team space without using a license:

- Configure IBM® Common License Key Server
- Modify the user roles in a team space
- Delete the team space



Remember: The floating license that was assigned to you is returned to the license pool when you perform any of the following actions:

- Log out from Rational® Test Automation Server.



Remember: The floating licenses assigned to prioritized floating users are not returned to the license pool when they log out from Rational® Test Automation Server.

- End a session on Rational® Test Automation Server or session is timed out.
- Complete the test runs on any of the third-party applications that are integrated with Rational® Test Automation Server.



Important:

Processor Value Unit (PVU) licenses are not supported for running Rational® Performance Tester schedules in Rational® Test Automation Server. You must configure floating licenses to run Rational® Performance Tester schedules in Rational® Test Automation Server.

Related information

[Configuring licenses for team spaces on page 139](#)

Configuring licenses for team spaces

As a *Team space Owner*, you can configure Rational License Key Server for the team spaces on the **License Configuration** page so that members of the team space can perform various operations such as creating a project, running tests, and so on.

Before you begin

You must have completed the following tasks:


- Configured the licenses that you purchased on Rational License Key Server.
- Installed the product and signed up as a user on Rational® Test Automation Server.
- Optional: Created a team space. See [Creating a team space on page 129](#).
- Been assigned the role of a *Team space Owner* of the team space.

About this task

You must have a valid license to work with the team spaces that you create on Rational® Test Automation Server. You must provide a separate IP address or Fully Qualified Domain Name (FQDN) of IBM® Common License Key Server for the team spaces when you configure the licenses. If you provided the IP address or FQDN of IBM® Common License Key Server that is already configured for any other team space, then the configuration of the license for the team space fails with an error message. If you delete the team space, then the IP address or FQDN of IBM® Common License Key Server used for that team space is released. You can then use that IP address or FQDN of IBM® Common License Key Server for another team space.



Note: When you upgrade the server software from the previous version to the latest version, then the license configuration of the previous release applies to the initial team space. You can verify the information from the **License Configuration** page of the initial team space.

1. Log in to Rational® Test Automation Server.
2. Click the **Settings** icon  from the left navigation pane.
3. Open the team space for which you want to configure the license by using the **Switch to** option.



Note: The **Switch to** option is displayed only if you have multiple team spaces.

4. Click the **Settings** icon , and then **License**.

Result

The **License Configuration** page is displayed.

5. **Optional:** Enter the number of floating user licenses in the **No. of floating users (Max)** field to limit the number of licenses accessed by users in the team space if you have a shared Rational License Key Server.



Note: The default selection is **Maximum**. Hence, the number of licenses is limited to the total number of licenses that are available in Rational License Key Server.

6. Enter the IP address or FQDN of the license server in the **Rational License Key Server** field to configure the licenses for the team space.



Note: You must enter either the IP address or FQDN in the *port_number@<IP_address> or <FQDN>* format. If you did not provide any port number, the default port number is used for Rational License Key Server.



For example, the value of **Rational License Key Server** can be:

- 27000@licenseserver.com
- 27000@127.0.0.1

You can view the following messages based on the value that you enter in the **Rational License Key Server** field:

Values	Messages
No value	Add the missing license details or enter the IP address or FQDN of License Server.
Incorrect format of IP address or FQDN	This is not a valid format of Rational License Key Server.
Correct value	License Server is already in use by another team space owner

7. Perform the following actions to view or hide the details of the License Server:

Fields	Icons	Action	Results
Rational License Key Server		Click	Rational® Test Automation Server shows the IP address or Fully Qualified Domain Name (FQDN) of Rational License Key Server that you have configured.
		Click	Rational® Test Automation Server hides the IP address or Fully Qualified Domain Name (FQDN) of Rational License Key Server that you have configured.

8. Click **Check connection** to verify the current status of Rational License Key Server.

Based on the availability of License Server, you can view the following messages:

- License Server is available
 - License Server is unreachable
9. Click **Configure**.
The **Licensing compliance** dialog is displayed.
 10. Click **Confirm** to apply the license configuration.

Results

You have configured the license for the team space. The **License usage** section displays the number of floating user licenses that are currently in use.



Remember: The **License usage** section does not display the number of **Concurrent Virtual Service** licenses that are currently in use.

What to do next

You can perform any of the following tasks:



- Add users to the **Prioritized Floating Users** list. See [Adding users to the Prioritized Floating Users list on page 141](#).
- Add members to your team space. See [Adding members to a team space on page 604](#).

Adding users to the Prioritized Floating Users list

As a *Team space Owner*, you can add users to the **Prioritized Floating Users** list so that the licenses are always available for those users.

Before you begin

You must have completed the following tasks:

- Been assigned the role of a *Team space Owner* of the team space.
 - Optional: Created a team space. See [Creating a team space on page 129](#).
 - Configured the license for the team space. See [Configuring licenses for team spaces on page 139](#)
1. Log in to Rational® Test Automation Server.
 2. Click the **Settings** icon  from the left navigation pane.
 3. Open the team space for which you want to add users from the **Switch to** option.
 4. Click the **Settings** icon , and then **License**.

Result

The **License Configuration** page is displayed.

5. Enter the name or the email ID of the user in the **Add prioritized floating users** field, and then select the name from the list that displays to add members to the **Prioritized Floating Users** list.



Note: You can press `Ctrl` to select multiple users simultaneously.

6. Click **Add** to make selected users as **Prioritized floating users**.

Results

You have added users to the **Prioritized Floating Users** list. The **License usage** section displays the following information:

- The number of floating user licenses that are currently in use
- The number of prioritized floating user licenses that are currently in use



Remember: The **License usage** section does not display the number of **Concurrent Virtual Service** licenses that are currently in use.



Removing or replacing prioritized users

When you want to remove a user from the list of prioritized users or you want to replace an existing user with another prioritized user, you as a *Team space Owner* can perform these changes from the **License Configuration** page.

Before you begin

You must have completed the following tasks:




- Been assigned the role of a *Team space Owner* for the team space that you want to modify the users from the **Prioritized Floating Users** list.
- Added users to the list of prioritized users for the team space on Rational® Test Automation Server.

1. Log in to Rational® Test Automation Server.
2. Click the **Settings** icon  from the left navigation pane.
3. Open the team space for which you want to remove or replace the users from the **Prioritized Floating Users** list from the **Switch to** option.
4. Click the **Settings** icon , and then **License**.

Result

The **License Configuration** page is displayed.

5. Perform the actions described in the following table to remove or replace users from the **Prioritized Floating Users** list:

Task	Actions
Remove a user from the list of prioritized users	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Locate the user that you want to remove from the Prioritized Floating Users list. Click the Menu icon  next to the name of the user, and then click Remove. Click Remove on the Remove prioritized user dialog to complete the action. <p> Remember: When you remove the user from the Prioritized Floating Users list, the test runs scheduled by the user fail to run if sufficient floating licenses are unavailable. If the user is currently on Rational® Test Automation Server, then the license can be used while the current session remains active.</p>
Replace a user in the list of prioritized users	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Locate the user from the Prioritized Floating Users list that you want to replace with another user. Click the Menu icon  next to the name of the user, and then click Replace. Enter the email ID or name of another user that you want to add to the Prioritized Floating Users list. Click Replace to complete the action.

Results

You have removed or replaced a user from the **Prioritized Floating Users** list.

Viewing license details of the team space

You can view license details of the team space to know information such as number of licenses that are configured, licensed users or details of the License Server on the **License Configuration** page.

Before you begin


The *Team space Owner* must have configured the licenses for the team spaces on Rational® Test Automation Server.

About this task

You as a licensed user can view only the number of licenses configured, details of the licensed users on the **License Configuration** page. However, if you are a *Team space Owner* or *Server Administrator*, you can also view the IP address or Fully Qualified Domain Name (FQDN) of Rational License Key Server that is configured for the team space.

1. Log in to Rational® Test Automation Server.
2. Perform one of the following actions described in the following table to view the license details of the team space:

Roles	Action
<i>Team space Owner</i> or <i>Server Administrator</i>	Perform Steps 3 on page 144 to 6 on page 144 .
Any other users	Perform Steps 3 on page 144 to 5 on page 144 .

3. Click the **Settings** icon  from the left navigation pane.
4. Open the team space for which you want to view the license details from the **Switch to** option.



Note: The **Switch to** option is displayed only if you have multiple team spaces.

5. Click the **Settings** icon , and then **License**.

Result



The **License Configuration** page is displayed with the following license information:

- The number of floating user licenses that are currently in use.
- The number of prioritized floating user licenses that are currently in use, if your *Team space Owner* has added the users to the **Prioritized Floating Users** list.
- The total number of floating user licenses that are configured by the *Team space Owner*.
- The name and email ID of the users that are configured as prioritized floating users by the *Team space Owner*.
- The status of License Server.



Remember: The **License usage** section does not display the number of **Concurrent Virtual Service** licenses that are currently in use.

6. Perform the following actions to view or hide the details of the License Server:

Fields	Icons	Action	Results
Rational License Key Server		Click	Rational® Test Automation Server shows the IP address or Fully Qualified Domain Name (FQDN) of Rational License Key Server that you have configured.
		Click	Rational® Test Automation Server hides the IP address or Fully Qualified Domain Name (FQDN) of Rational License Key Server that you have configured.

Results

You have viewed the details about the licenses for the team space.

Chapter 5. Test Author Guide

This guide describes how to create test assets in Rational® Test Automation Server and publish test assets to the Git repository that you have configured. You can also edit existing test assets that are in the repository added to your project. This guide is intended for testers and test managers.


Edit branch overview

When you want to create or modify dataset in Rational® Test Automation Server, you must use an Edit branch.

You must be a *Team Space Owner, Project Owner, or Member* with a Tester role to create or edit dataset. You can create the Edit branch during the creation or modification of datasets. You can create or select the Edit branch from Rational® Test Automation Server as shown in the following table:

Rational® Test Automation Server UI	Actions			
	Dataset creation	Dataset modification	Edit branch creation	Edit branch selection
Author > Datasets	Yes	Yes	Yes	Yes
Author > Changes	No	Yes	No	Yes

While you provide a name for the Edit branch, you must adhere to the rules of Git branch naming conventions. For more information, refer to the [Git](#) documentation. When you create the Edit branch, any member of the project can use your Edit branch while the branch is available. You can also view all the Edit branches from the **Branch** drop-down list when other members of the project create Edit branches.

To differentiate the remote branch and Edit branch from the **Branch** drop-down list, an edit icon  and the initials of the member who created the Edit branch are shown for the Edit branch. After you create the Edit branch, you can also view the Edit branch by accessing the remote Git repository from a browser.

After you create or edit the test assets, you can either publish the test assets to a remote Git repository or discard the changes made to the test assets. All created or modified test assets reside in the Edit branch until you publish or discard the changes made to the test assets.

The Edit branch automatically disappears from the **Branch** drop-down list and remote Git repository when you perform any of the following actions:

- Publish changes to the remote Git repository
- Discard the changes made in all test assets




Note: After the Edit branch disappears, the **Branch** field is set to the previously selected branch name.

Datasets overview

A dataset provides tests with variable data during a run. The test that uses a dataset at run time replaces a value in the recorded test with variable test data that is stored in the dataset.

In Rational® Test Automation Server, you can create a dataset and use it to replace the dataset values with original values during run time, when you want to run test assets that contain the dataset.

You can perform the following actions from **Author > Datasets**:

Actions	Descriptions
View	You can view the contents of a dataset. See Viewing a dataset on page 163 .
Create	You can create a dataset to use it during a test or schedule run. See Creating a dataset on page 148 .
Edit	You can edit a dataset when you want to run a test asset with different dataset values. See Editing a dataset on page 152 .
Delete	You can delete the dataset when it is not required in your test environment. See Deleting a dataset on page 164 .
Save As	You can save a copy of a dataset in a different folder or make a copy with a different name by clicking the Menu icon  and then by selecting the Save As option.
Branch selection	You can view the dataset listed in the other branches of the repository by selecting the name of the branch from the Branch list. When you access the Datasets page for the first time after adding the repository, the global branch is set to the default branch of the repository. When multiple repositories are added to the same project, the following events occur:

Actions	Descriptions
	<ul style="list-style-type: none"> • The datasets stored in the selected branch of all the repositories are displayed. • All the branches in all the repositories are listed in the drop-down list. • If the selected branch is not present in a repository, then datasets from that repository are not displayed.

**Notes:**

- Only a project **Owner** or a project member whose role is a **Tester** can view and edit the dataset.
- To view the content of an encrypted dataset, you must provide an encryption key that you set while encrypting the dataset column.
- As a **Viewer**, when you try to access the **Datasets** page, an error message is displayed because you do not have the permission to view this page.

You can create a dataset in Rational® Test Automation Server by using any of the following method:

- Empty dataset
- CSV file

Creating a dataset

You can create datasets in Rational® Test Automation Server to replace the values of existing dataset with the values of new dataset during a test or schedule run.

Before you begin

You must be a member of the project with the **Owner** or **Tester** role, and you must have completed the following tasks:

- Created a project in Rational® Test Automation Server. See [Adding a project on page 619](#).
- Added a Git repository to your project. See [Adding repositories to a server project on page 620](#).

About this task

The default dimension of the dataset that you create in Rational® Test Automation Server is 1 Row, 1 Column (1R X 1C) dataset. You can enter the number of columns and rows required for the dataset during the creation of dataset.

When you want to create or modify any dataset, you must have an edit branch. You can create the edit branch during the creation of dataset. All your dataset reside in the edit branch until you publish or discard the changes.

After you create the dataset, you can either publish your changes to a remote Git repository or you can discard the changes made to the dataset. The edit branch automatically disappears from the **Branch** list as well as from the remote Git repository and sets the global branch to the previously selected branch in the following conditions:

- When you publish changes to the remote Git repository.
- When you discard the changes made in the dataset.

1. Log in to Rational® Test Automation Server if you are not already logged in.
2. Open a project from the team space.
3. Go to **Author > Datasets**, and then select the **Branch** from the list where you want to store the dataset.
4. Click **Create dataset**.

Based on the **Branch** that you selected, the following window is displayed:

- **Edit Branch** - If you selected other than the edit branch.
- **Create dataset** – If you selected the edit branch.

5. Perform one of the steps described in the following table to select or create the edit branch:

Step description	Step no
To select the edit branch	Perform 5.a on page 149
To create the edit branch	Perform 5.b on page 149

- a. Perform the following steps to select the edit branch:

- i. Choose the edit branch from the **Select branch** drop-down list if the edit branches are already existing.
- ii. Click **Ok**.

- b. Perform the following steps to create the edit branch:

- i. Select **Create new branch**.
- ii. Provide a name for the edit branch.
- iii. Click **Save**.

6. Enter a name for the dataset in the **Asset name** field.
7. Select the place where you want to save the dataset in the **Location** field, and then click **Next**.
8. Choose **Create an empty dataset** from the **Select source** list.
9. Enter the number of columns and rows required for the dataset that you want to create.
10. Click **Create**.

Result

The new dataset opens in a CSV Editor in a web browser. The dataset created is listed on the **Datasets** page.

Results

You have created the dataset in your project.

What to do next

You can perform the following tasks:

- Add, modify, or remove data in the dataset. See [Editing a dataset on page 152](#).
- Publish the dataset to the Git repository so that other members of the project can use the dataset. See [Publishing a dataset on page 162](#).

Creating a dataset by using a CSV file

When a CSV file contains a large amount of data, you can use such CSV file to create a dataset in Rational® Test Automation Server.

Before you begin

You must be a member of the project with the **Owner** or **Tester** role, and you must have completed the following tasks:

- Created a project in Rational® Test Automation Server. See [Adding a project on page 619](#).
- Added a Git repository to your project. See [Adding repositories to a server project on page 620](#).
- Saved a CSV file that contains variable data on your computer.

About this task

When you want to create or modify any dataset, you must have an edit branch. You can create the edit branch during the creation of dataset. All your dataset reside in the edit branch until you publish or discard the changes.

After you create the dataset, you can either publish your changes to a remote Git repository or you can discard the changes made to the dataset. The edit branch automatically disappears from the **Branch** list as well as from the remote Git repository and sets the global branch to the previously selected branch in the following conditions:

- When you publish changes to the remote Git repository.
- When you discard the changes made in the dataset.

1. Log in to Rational® Test Automation Server if you are not already logged in.
2. Open a project from the team space.
3. Go to **Author > Datasets**, and then select the **Branch** from the list where you want to store the dataset.
4. Click **Create dataset**.

Based on the **Branch** that you selected, the following window is displayed:

- **Edit Branch** - If you selected other than the edit branch.
- **Create dataset** – If you selected the edit branch.

5. Perform one of the steps described in the following table to select or create the edit branch:

Step description	Step no
To select the edit branch	Perform 5.a on page 151
To create the edit branch	Perform 5.b on page 151

a. Perform the following steps to select the edit branch:

- i. Choose the edit branch from the **Select branch** drop-down list if the edit branches are already existing.
- ii. Click **Ok**.

b. Perform the following steps to create the edit branch:

- i. Select **Create new branch**.
- ii. Provide a name for the edit branch.
- iii. Click **Save**.

6. Enter a name for the dataset in the **Asset name** field.

7. Select the place where you want to save the dataset in the **Location** field, and then click **Next**.

8. Choose **Import from a CSV file** from the **Select source** list.

9. Click **Select File** and choose the CSV file that contains variable data to import into the dataset.

10. Enable **First row contains headers** by using the toggle icon if the CSV file contains the header.

11. Click **Create**.

Result

The new dataset opens in a CSV Editor in a web browser. The dataset created is listed on the **Datasets** page.

Results

You have created the dataset by using the CSV file in your project.

What to do next

You can perform the following tasks:

- Add, modify, or remove data in the dataset. See [Editing a dataset on page 152](#).
- Publish the dataset to the Git repository so that other members of the project can use the dataset. See [Publishing a dataset on page 162](#).

Editing a dataset

You can add, modify, remove, import, or export data from a dataset by using the CSV Editor. The working principle of the CSV Editor is similar to that of a spreadsheet.

Before you begin

You must have created a dataset or configured a repository that contains the dataset.

About this task




If you are a project **Owner** or **Tester** in Rational® Test Automation Server 10.1.0 or later, you can perform basic tasks in the CSV Editor by right-clicking any row, column, or cell in the dataset to organize your data in a better way. For example, you can perform tasks such as updating data in a cell, inserting or deleting rows and columns, or renaming column names.

When you edit the dataset in the CSV Editor, you can use the following keyboard shortcuts to control the cursor selection in the CSV Editor:

- **Tab** - To move the cursor control to the next available option.
- **Shift-Tab** – To move the cursor control to the previous option.
- **Shift+F10** – To open the context menu from the dataset cell.

After you edit the dataset, you can save the changes made to the dataset, and then you can publish the dataset to the Git repository. If you save and close the edited dataset, the **Changes** page lists the edited dataset and later you can publish to the Git repository for other members to use.

When you or other members of the project edit the same dataset, you can view an icon with the initials of the member next to the name of the dataset on the **Datasets** and **Changes** pages. If you do not view the icon, you must refresh the **Datasets** page to view the icons. The **Changes** page denotes conflicting edits when another member publishes their edited dataset first.

Dataset name	Table matrix	Actions
> AdditionalLocations AS JD <small>LogData/Logical/Locations/AdditionalLo</small>	2 Col × 7 Row	  
> Locations <small>LogData/Logical/Locations/Locations.sit</small>	3 Col × 3 Row	 

When a member with conflicting edits tries to publish the edited dataset, an error about conflicting changes is displayed. However, the member can use the **Save As** option to save and publish a copy of the dataset edits under a new asset name. The member must discard the edits that were made to the original dataset.

For example, consider a scenario when Amy Smith and John Doe edit the same dataset, and Amy Smith edits and publishes the dataset. On the **Changes** page, a message that indicates number of commits ahead and behind in the branch is displayed when John Doe tries to publish the same dataset.



Note: You cannot resize the width of rows in the CSV Editor. When you have a large amount of data in a cell, you can right-click the cell and select **Copy** (or Ctrl+C), and then paste it into a text-editing program to view the content. Alternatively, you can hover the mouse over the cell to view the content.

When you have a CSV file that has data separated from a character, then you can import that CSV file into the dataset. You can select any of the following separator characters from the **Configure Dataset** window, and the selection can be the separator character that you used in the CSV file:

- Comma
- Semicolon
- Space
- Tab
- Other

Consider that you have the data in the CSV file in the following format:

Name;CCNum
John;1234 5678 1234 5678
Bob;1122 3344 5566 7788
Amy;2233 4455 6677 8899


When you import the CSV file in the dataset, and then select the separator value as **Semicolon**, the data in the dataset is displayed as follows:

	Name	CCNum
1	John	1234 5678 1234 5678
2	Bob	1122 3344 5566 7788
3	Amy	2233 4455 6677 8899

If you want the data in its original format, that is, a semicolon (;) character to separate the data, then you can choose any other separator value from the **Configure Dataset** window.



Note: The default separator value is **Comma**.

1. Log in to Rational® Test Automation Server if you are not already logged in.
2. Open a project from the team space.
3. Go to **Author > Datasets**, and then select the **Branch** from the list to find the dataset which you want to edit.
4. Click the **Edit** icon  from the **Actions** column of the dataset.

Based on the **Branch** that you selected, the following events occur:

- The **Edit Branch** window is displayed if you selected other than the edit branch.
 - The dataset opens in the CSV Editor in a web browser if you selected the edit branch.
5. Perform one of the steps described in the following table to select or create the edit branch:

Step description	Step no
To select the edit branch	Perform 5.a on page 154
To create the edit branch	Perform 5.b on page 154



a. Perform the following steps to select the edit branch:






- i. Choose the edit branch from the **Select branch** drop-down list if the edit branches are already existing.
- ii. Click **Ok**.





b. Perform the following steps to create the edit branch:



- i. Select **Create new branch**.
- ii. Provide a name for the edit branch.
- iii. Click **Save**.



6. Perform the following actions to use the options available in the CSV Editor:



Options	Actions
Find and Replace 	To find: <ol style="list-style-type: none"> a. Click the Find and Replace icon . b. Enter the content that you want to search in the Find field. c. Select any or all the following options to find the search content more effectively:

Options	Actions
	<ul style="list-style-type: none"> ▪ Select the Case sensitive checkbox to search the content that is the exact letter case of the content entered in the Find field. ▪ Select the Match entire cell contents checkbox to search for cells that contain only the characters that you have entered in the Find field. ▪ Select the Search using regular expression checkbox to search the pattern that matches strings. <p>For example, to search a cell that contains any number between 0 to 9, do the following:</p> <ol style="list-style-type: none"> i. Enter <code>\d</code> in the Find field. ii. Select the Search using regular expression checkbox. iii. Click Find. <p>d. Click Find. If the text is found, the cell containing that text is selected.</p> <p>e. Click Find again to find further instances of the search text.</p> <p>To find and replace:</p> <ol style="list-style-type: none"> a. Click the Find and Replace icon . b. Enter the content that you want to search in the Find field. c. Enter the content that you want to replace in the Replace field. d. Select any or all the following options to find and replace the content more effectively: <ul style="list-style-type: none"> ▪ Select the Case sensitive checkbox to find the content that is the exact letter case of the content entered in the Find field. ▪ Select the Match entire cell contents checkbox to find and replace for cells that contain only the characters that you have entered in the Find and Replace fields. ▪ Select the Search using regular expression checkbox to find and replace the pattern that matches strings. e. Click Replace to replace the individual instances. f. Click Replace All to replace every instance of the content throughout the dataset.
Un- do 	<ol style="list-style-type: none"> a. Click the Undo icon . b. Select the recent changes from the list that you want to undo, and then click the list. <p>The Undo option undoes anything you do in the dataset. The CSV Editor saves the unlimited undo-able action. You can perform the undo action even after you save your changes made to the dataset.</p>
Redo 	<ol style="list-style-type: none"> a. Click the Redo icon . b. Select the recent changes from the list that you want to redo, and then click the list. <p>The CSV Editor saves the unlimited redo action.</p>

Options	Actions
Import 	<p>When there are large amount of data stored in a CSV file, you can import variable data into a dataset.</p> <p>Importing data from a CSV file</p> <ol style="list-style-type: none"> Click the Import icon . Select Import from CSV a file from the drop-down list. Click Select file and choose the CSV file that contains variable data to import into the dataset. Enable the First row contains headers by using the toggle icon if your CSV file contains the header. Select one of the following options to append or overwrite data in the dataset: <ul style="list-style-type: none"> ▪ Enable Overwrite existing data by using the toggle icon to add the rows and columns from the beginning of the dataset. ▪ Enable Append existing data by using the toggle icon to add rows and columns to the end of the dataset. Click Import.
Export 	<p>You can export variable data from the dataset into a CSV file to reuse in future tests when required. You must have a dataset that you want to export.</p> <p>Click the Export icon  to download the dataset as a CSV file.</p>
Set as current row	<p>During the test run, if you want variable data to be selected from a current row instead of the first row in a dataset, right-click any cell in a row and select Set as current row.</p> <p>Also, you can set the current row from the Datasets page by clicking Menu, and then the Configure option.</p> <p>When rows are deleted:</p> <p>If you delete any row between row 1 to current row, the current row data is taken from the next row. For example, when you set the current row as 6, and then you delete any row between row 1 to row 6, the current row remains at row 6, but the content of row 7 is moved to row 6.</p> <p>When rows are inserted:</p>

Options	Actions
	<p>If you insert any new row between row 1 to the current row, the current row data is taken from the previous row. For example, when you set the current row as 6, and then you insert any row between row 1 to row 6, the current row remains at row 6, but the content of row 5 is moved to row 6.</p>
Dataset Configuration 	<p>In the Configure Dataset window, you can change the row and column settings and configure the string values in the dataset that contains variable data for tests to use when they run.</p> <ol style="list-style-type: none"> a. Click the Menu icon  , and then select the Configure option. b. Select any of the separator values that you used in the CSV file. <p>The available options are Comma, Semicolon, Space, Tab, and Other. In the CSV file, if you have any other separator characters other than the available options, then you can select the Other option, and then can specify a value.</p> <p>For example, if the data in the CSV file is separated by a character #, then select the Other option and enter # in the field.</p> c. Configure the following options to change the row and column settings: <ul style="list-style-type: none"> ▪ Column header - Use an up-down control button to increment or decrement the value of the column header. ▪ Data start point - Use an up-down control button to increment or decrement the value of the data starting pointer. ▪ Current row - Use an up-down control button to increment or decrement the value of the current row. d. Configure the following options to change the string values in the dataset: <ul style="list-style-type: none"> ▪ Treat as null - Enter a string value that is to be treated as null when running the test. ▪ Treat as empty - Enter a string value that is to be treated as empty when running the test. <p>For example, when you run the test and the data 123 in the dataset to be treated as empty, then you can specify 123 in the Treat as empty field.</p> ▪ Treat empty text as null - Select this field when you want the dataset that contains any blank cells, and the value of those blank cells to be interpreted as null. e. Click Update to apply the changes.

Options	Actions
Discard 	Click the Menu icon  and select Discard to discard the changes made to the dataset.

7. Click the **Save** icon  to save the changes made to the dataset.
8. Click the **Publish** icon  to publish the dataset to the Git repository, and then close the CSV editor.

Results

You have edited the dataset.

Dataset encryption

Encrypted datasets are useful when you want to run tests that contain confidential information such as a set of passwords or account numbers.

When you run a test that uses an encrypted dataset, then you must provide an encryption key for decrypting the encrypted data in columns so that the data can be used in the test. If the test uses data from the multiple encrypted dataset columns, you must enter the same encryption key for every encrypted dataset column that the test uses.

When you run the test that uses the dataset with the encrypted column, the value of the column is decrypted at a run time. The data in the column is sent as a clear-text string in requests to the server. The actual values of the encrypted dataset variables are not displayed in the test log. The test log displays asterisks for the encrypted dataset variables.

You can use only one encryption key to encrypt data in the columns in any dataset.

Important:

The encryption keys that you use to encrypt data in a dataset are not stored on the server nor can be retrieved from the server. Therefore, you must remember to store the encryption keys in a secure location. You must use the same encryption keys to perform the following operations:


- View the encrypted values
- Decrypt data
- Enable the use of the encrypted dataset during test runs

Encrypting a dataset column

To secure test data, you must encrypt datasets. You can encrypt data in the columns of a dataset by using an encryption key. When you run a test that utilizes a dataset with encrypted variables, you must enter the encryption key for the encrypted column that the test uses.

Before you begin

You must have created a dataset. See [Creating a dataset on page 148](#).

1. Log in to Rational® Test Automation Server if you are not already logged in.
2. Open a project from the team space.
3. Go to **Author > Datasets**, and then select the **Branch** from the list to find the dataset that you want to encrypt.
4. Click the **Edit** icon  from the **Actions** column of the dataset.

Based on the **Branch** that you selected, the following events occur:

- The **Edit Branch** window is displayed if you selected other than the edit branch.
 - The dataset opens in the CSV Editor in a web browser if you selected the edit branch.
5. Perform one of the steps described in the following table to select or create the edit branch:

Step description	Step no
To select the edit branch	Perform 5.a on page 159
To create the edit branch	Perform 5.b on page 159

- a. Perform the following steps to select the edit branch:
 - i. Choose the edit branch from the **Select branch** drop-down list if the edit branches are already existing.
 - ii. Click **Ok**.
 - b. Perform the following steps to create the edit branch:
 - i. Select **Create new branch**.
 - ii. Provide a name for the edit branch.
 - iii. Click **Save**.
6. Right-click any cell in a column that you want to encrypt and select **Encrypt column data**.

Result

The **Encrypt Column** window is displayed.

7. Enter an encryption key in the **Encryption Key** field to encrypt the data in the column.



Remember: When you have already encrypted other columns in the dataset, you must enter the same encryption key that you used previously. You can use only one encryption key to encrypt columns in a dataset.



Important:

The encryption keys that you use to encrypt data in a dataset are not stored on the server nor can be retrieved from the server. Therefore, you must remember to store the encryption keys in a secure location. You must use the same encryption keys to perform the following operations:

- View the encrypted values
- Decrypt data
- Enable the use of the encrypted dataset during test runs

8. Click **Encrypt Column**.

Result

Asterisks are displayed instead of actual data for the encrypted column.

Results

You have encrypted the dataset column in your project.

What to do next


You can publish the dataset to the Git repository so that other members of the project can use the dataset. See [Publishing a dataset on page 162](#).

Decrypting a dataset column

To view the content of an encrypted dataset, you can decrypt the dataset column. Removing encryption from a dataset revokes the protection offered to the test data.

Before you begin

You must have created at least one dataset and encrypted the dataset with an encryption key. See [Creating a dataset on page 148](#) and [Encrypting a dataset column on page 158](#).

1. Log in to Rational® Test Automation Server if you are not already logged in.
2. Open a project from the team space.
3. Go to **Author > Datasets**, and then select the **Branch** from the list to find the dataset that you want to decrypt.
4. Click the **Edit** icon  from the **Actions** column of the dataset.

Based on the **Branch** that you selected, the following events occur:

- The **Edit Branch** window is displayed if you selected other than the edit branch.
 - The dataset opens in the CSV Editor in a web browser if you selected the edit branch.
5. Perform one of the steps described in the following table to select or create the edit branch:

Step description	Step no
To select the edit branch	Perform 5.a on page 161
To create the edit branch	Perform 5.b on page 161

- a. Perform the following steps to select the edit branch:
 - i. Choose the edit branch from the **Select branch** drop-down list if the edit branches are already existing.
 - ii. Click **Ok**.
 - b. Perform the following steps to create the edit branch:
 - i. Select **Create new branch**.
 - ii. Provide a name for the edit branch.
 - iii. Click **Save**.
6. Right-click encrypted cells that display the contents with asterisks, and then select **Decrypt column data**.
- Result**
The **Decrypt Column** window is displayed.
7. Enter the encryption key that you used to encrypt the data in the column in the **Encryption Key** field.
8. Click **Decrypt Column**.

Result

Asterisks are replaced with the actual data in the decrypted column.

Results

The encryption is removed from the selected column in the dataset. When you run a test that uses a dataset that contains decrypted data, the variable data is substituted for the original data in the recorded test without prompting for the encryption.

What to do next

You can publish the dataset to the Git repository so that other members of the project can use the dataset. See [Publishing a dataset on page 162](#).

Publishing a dataset

After you create or edit any datasets in Rational® Test Automation Server, you can publish your changes to a remote Git repository. Therefore, after you publish the dataset, other members in the project can use your dataset in their test run, if required. The Edit branch automatically disappears from the **Branch** drop-down list and changes made to the datasets are merged into the previously selected branch name.

Before you begin


You must have created, edited, or deleted dataset assets in Rational® Test Automation Server.

About this task

When you create or edit multiple datasets, the **Changes** page lists all datasets in the corresponding Edit branches. When you have many datasets listed on the **Changes** page, you cannot publish or discard a single dataset. You must either publish all datasets or discard all changes made to datasets. If you do not modify any datasets, the **Changes** page displays a message that indicates there are no changes in any of the datasets.

The **Changes** page also provides you with information about the number of commits that the Edit branch is ahead and behind. You can click the **(n) commits ahead** option, and then click **Reset** to revert your changes. Similarly, you can also click the **(n) commits behind** option, and then click **Merge** to merge the changes of other member into the Edit branch.

You can publish the dataset to the remote Git repository either from the **Changes** page or from the CSV Editor. When you modify the dataset and publish it to the Git repository, the other members who have access to that dataset can view the updated dataset.

You can click the **Menu** icon  to perform the following actions from the **Changes** page:

Options	Actions
Edit	Click Edit to make any updates to the dataset before publishing.
Discard	Click Discard to remove the changes that you made to the dataset asset.
Save As	Click Save As to save a copy of the dataset.

1. Log in to Rational® Test Automation Server if you are not already logged in.
2. Open a project from the team space.
3. Go to **Author > Changes**.
4. Click **Publish changes**.
5. Enter a description for the changes made to the dataset, and then click **Publish**.

Result

A notification is displayed for successful pushing of changes to the Git repository.

Results


You have published the modified dataset into the Git repository.

Viewing a dataset

When test assets includes datasets, you can view the contents of a dataset from Rational® Test Automation Server. The datasets residing in the Git repository are listed in the **Datasets** page.

Before you begin

You must be a member of the project with the **Owner** or **Tester** role, and you must have completed the following tasks:

- Created a project in Rational® Test Automation Server. See [Adding a project on page 619](#).
 - Added a Git repository to your project. See [Adding repositories to a server project on page 620](#).
 - Created at least one dataset and encrypted the dataset with an encryption key. See [Creating a dataset on page 148](#) and [Encrypting a dataset column on page 158](#).
 - Created a classification for an encrypted dataset. See [Creating a classification on page 647](#).
1. Log in to Rational® Test Automation Server if you are not already logged in.
 2. Open a project from the team space.
 3. Go to **Author > Datasets**, and then select the **Branch** from the list.
 4. Find the dataset that you are interested in, and then expand the dataset by clicking the **Expand** icon .





Note: You can click the **Dataset name** field to sort the datasets by name in alphabetical order. Alternatively, you can use the **Search** field to search the dataset by name.

Results

You have viewed the contents of the dataset.

Viewing an encrypted dataset

You can use the **Dataset** page to view the contents of an encrypted dataset from Rational® Test Automation Server.

1. Go to the **Datasets** page, find the encrypted dataset that you are interested in, and then expand the dataset by clicking the **Expand** icon .
2. Perform the following steps, if you have created a classification but yet to add the encrypted dataset to it:
 - a. Find the encrypted dataset that you are interested in, and then click the **Lock** icon  from the **Actions** column of the dataset.
 - b. In the **Change the classification for the dataset** window, select a classification from the list and enter the encryption key set for the dataset.
 - c. Click **Save** to save the classification details.
3. Right-click the encrypted column and click **Show encrypted data**.

You can right-click the decrypted column and select **Hide encrypted data** to encrypt the data again.



Important: The **Show encrypted data** option is available only when you have added the encrypted dataset to the respective classification.

Results


You have viewed the contents of an encrypted dataset.

Changing classification for an encrypted dataset

To move the encrypted dataset from one classification to another, you can change the classification for an encrypted dataset from the **Dataset** page.

Before you begin

You must have created at least two or more classifications. See [Creating a classification on page 647](#).

1. Go to the **Datasets** page and find an encrypted dataset for which you want to change the classification.
2. Click the **Lock** icon  from the **Actions** column of a dataset.
3. In the **Change the classification for the Dataset** window:
 - a. Choose the classification from the list.
 - b. Enter the encryption key for the dataset that was set while encrypting the dataset column.
 - c. **Save** the classification details.



Remember: You can perform this task only if you know the encryption key set for the dataset while encrypting the dataset column.

Results

You have changed the classification for an encrypted dataset.

Related information

[Managing an encrypted dataset on page 646](#)

[Creating a classification on page 647](#)

Deleting a dataset

You can delete the dataset when it is not required in your test environment.

Before you begin

You must have at least one dataset asset in your Git repository that you have configured.


About this task

You can delete datasets in Rational® Test Automation Server in the following scenarios:

- Datasets that you created in the desktop clients and that are cloned to the repository in your project on Rational® Test Automation Server.
- Datasets that you created in Rational® Test Automation Server and that are published to the repository in your project.
- Datasets that are in the `.csv` file format.

You cannot delete the following datasets:

- Datasets that you created or edited in Rational® Test Automation Server and that are not published to the repository in your project.
- Datasets that are in the `.sit` file format.

1. Log in to Rational® Test Automation Server if you are not already logged in.
2. Open a project from the team space.
3. Go to **Author > Datasets**, and then select the **Branch** from the list to find the dataset that you want to delete.
4. Click the **Menu** icon  from the **Actions** column of a dataset, and then click **Delete**.

Based on the **Branch** that you selected, the following events occur:

- The **Edit Branch** window is displayed if you selected other than the edit branch.
 - The **Delete dataset** window is displayed if you selected the edit branch.
5. Perform one of the steps described in the following table to select or create the edit branch:

Step description	Step no
To select the edit branch	Perform 5.a on page 165
To create the edit branch	Perform 5.b on page 165

- a. Perform the following steps to select the edit branch:

- i. Choose the edit branch from the **Select branch** drop-down list if the edit branches are already existing.
- ii. Click **Ok**.

- b. Perform the following steps to create the edit branch:

- i. Select **Create new branch**.
- ii. Provide a name for the edit branch.
- iii. Click **Save**.

6. Perform any of the following actions to decide from where the dataset is to be deleted:

Choose from:

- Select the **Publish deletion from <branch name of the Git repository> to <branch name of the Git repository>** checkbox to delete the dataset from the remote Git repository.
- Clear the **Publish deletion from <branch name of the Git repository> to <branch name of the Git repository>** checkbox to delete the dataset from the project.



Note: When you clear the **Publish deletion from <branch name of the Git repository> to <branch name of the Git repository>** option, the deletion of the dataset does not reflect in the remote Git repository. Therefore, the other members of the project can still use the dataset that you have deleted. If you want, later you can publish the dataset to the Git repository from the **Changes** page.

7. Perform the following actions to delete the dataset:

Option	When	Action
Publish deletion from <branch name of the Git repository> to <branch name of the Git repository>	Selected <input checked="" type="checkbox"/>	a. Enter a description for deleting the dataset in the Description of change field. b. Click Delete and Publish .
	Cleared <input type="checkbox"/>	Click Delete .

Results

You have deleted the dataset.

Chapter 6. Test Execution Specialist Guide

This guide describes tasks that you must complete before you can configure and run tests in Rational® Test Automation Server. You can find information about configuring runs for the different test types that are supported. You can also find information about other tasks that you can perform on the Resource Monitoring Service, Virtualization, and Integrations with third-party applications. This guide is intended for testers or test execution specialists.

System modeling

You can use the system model feature on Rational® Test Automation Server to create a model that simulates an application or system under test. You can define components to represent the actual components in the system under test, define their relationships and then view the components in a system under test as a diagrammatic representation.

Introduction to a system model

A system model is a logical presentation of the components contained in a system under test and the relationships between components. You can create a system model when you want to visualize the different components and their relationships as a diagrammatic representation.

You can create only one system model within a team space on Rational® Test Automation Server. A team space repository is required to contain the system model that you create. Before you can create a system model, you must add a team space repository. The system model that is created or edited is stored in the team space repository. All projects within a team space share the same system model. All members of a team space can create, edit, and view the system model.

After you log in to Rational® Test Automation Server, you can go to the **System Model** page to create a system model for your team space. Alternatively, you can go to any of the projects in your team space and create a system model from the **System Model** page within a project. The system model is accessible from all the projects in your team space.

Components in a system model

Components are the basic building blocks that are used to represent a piece of software in an application.

You can model a system by using components. You can create components that represent the type of asset or resource used by the system or application under test and can be of the following types:

Component type	Description
Database	The component type represents the database resource. You can select this component type when you want to represent a database asset or resource in the system model.

Component type	Description
Service	The component type represents the virtual service resource. You can select this component type in the system model to represent a virtual service in the team space repository or associate a virtual service resource that is in the project repository.
UI	The component type represents the user interface (UI) resource. You can select this component type when you want to represent the UI resource in the system model.

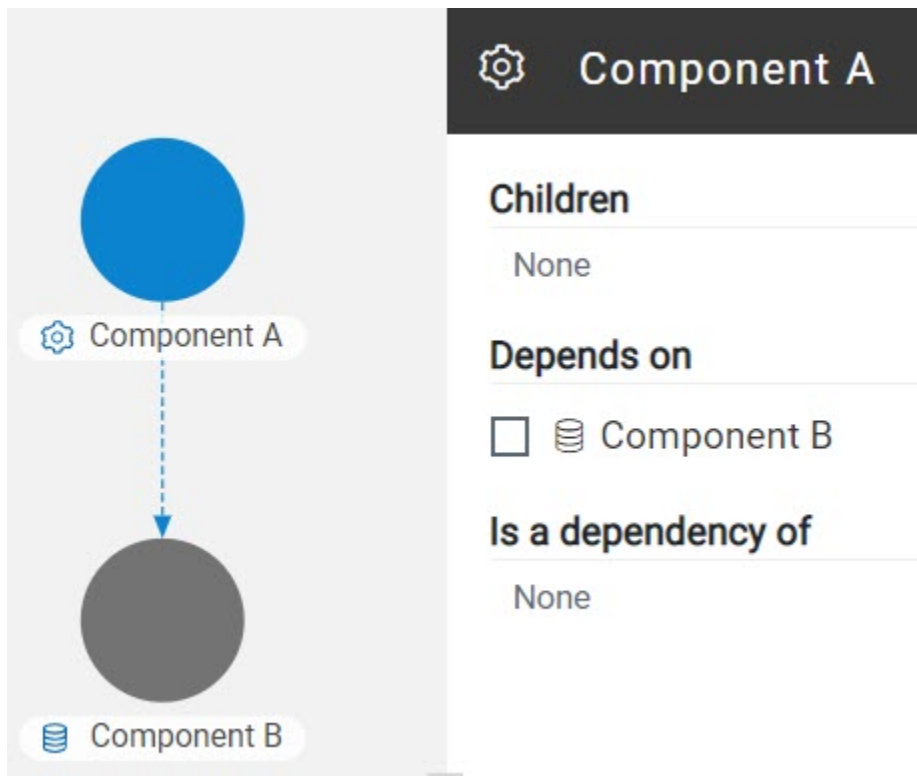
Component relationships

You can create multiple components to depict the different resources in the system or application under test. You can create a component as a *child* component of a *parent* component. You can also create multiple components as children components of the same parent component.

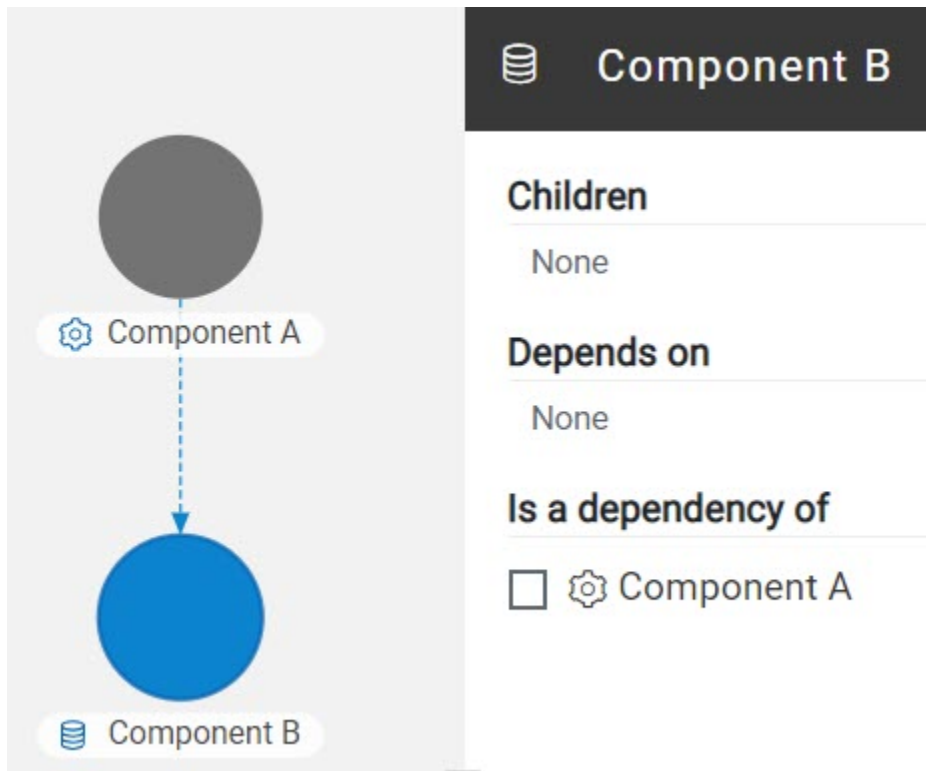
You can define the relationship between components if one component depends on another or if a component has a dependency of another component. The **Depends on** or **Is a dependency of** relationship depends on the component that you select in the system model after you define the relation.

For example, if you define *Component A* to depends on *Component B*, then the relationship is displayed with a line arrow from *Component A* to *Component B*, and the arrow points to *Component B*.

When you click the *Component A* in the system model page, you can view that the *Component B* is listed under the **Depends on** section for *Component A* in the right pane.



When you click the *Component B* in the system model, you can view that the *Component A* is listed under the **Is a dependency of** section for *Component B* in the right pane.





Component associations







When you work with the system model from your project in a team space, you can associate test resources that are in your project repository with components in the system model. You can associate all types of test resources such as Suites or tests, and virtual service resources with the components.

After associating test resources, you can view them when you select the component with which they are associated. You can select the associated test resources to start a run of the Suites or tests or start an instance of the virtual services resource.




Using the system model UI

The following table lists the icons used in a system model and the icons that are displayed on the **System Model** page:



Icon	Description
	Represents a system model.
	Represents Suites and tests in a repository.





Icon	Description
	Represents a database resource. The component name is prefixed with this icon when you select the component type as database.
	Represents a service resource. The component name is prefixed with this icon when you select the component type as service.
	Represents a user interface (UI) resource. The component name is prefixed with this icon when you select the component type as UI.
	Represents the virtual services.
	Represents the dependency between the components.
	Represents the inherited dependencies.

The following table lists the icon that you can use to perform different actions in the system model:

Icon	Used for...
	Adding components, child components, or dependent components.
	Searching for components in the system model.
	Publishing the changes made to the system model, changes made to components, or changes made to component associations. The changes are published to the team space repository.

You can perform the following operations when you view the model that is displayed on the **System Model** page:

Operation	Action
Pan the view of the model	<ol style="list-style-type: none"> 1. Click the area of the model that you want to pan and move the area by keeping the left mouse button pressed. 2. Release the left mouse button to stay on the view that you panned.
Zoom in the view of the model	<ol style="list-style-type: none"> 1. Click the Zoom-in icon . <p>The view is enlarged or zoomed in.</p> <p> Note: You can repeatedly click the Zoom-in icon to further enlarge the view.</p> <p>Alternatively, you can use the scroll button on the mouse or device to zoom in the view.</p>

Operation	Action
Zoom out the view of the model	<p>1. Click the Zoom-out icon .</p> <p>The view is diminished or zoomed out.</p> <p> Note: You can repeatedly click the Zoom-out icon to further diminish the view.</p> <p>Alternatively, you can use the scroll button on the mouse or device to zoom out the view.</p>
Reset the view	<p>1. Click the Menu icon  on the System Model page.</p> <p>2. Select Reset Zoom.</p> <p>The view is reset to the default view without any zoom applied. If the view was zoomed in or zoomed out, the view is reset to 100% of the size.</p>
Position the model view	<p>1. Click the Menu icon  on the System Model page.</p> <p>2. Select from the following options:</p> <ul style="list-style-type: none"> ◦ Center: Select this option to align the view of the model to the center of the pane. ◦ Fit to View: Select this option to fit the view of the model in the pane on the System Model page.

Tasks in system modeling

When you want to model the system under test for the first time after you install the server software, you must first add the team space repository. See [Adding a repository to a team space on page 173](#).

After a repository is configured as the team space repository, you can work with the system model. See [Tasks for working with a system model on page 172](#).

When you want to change, update or delete the repository, you can work with the team space repository. See [Tasks for working with the team space repository on page 173](#).

Task flows for working with a system model

You can find information about the tasks that you can perform when you want to create a system model, components, associate the components, create linkages among the components, and then publish the system model

to the team space repository. You can also find information about the tasks that you must perform when you want to modify an existing system model.

Tasks for working with a system model

You can find the tasks when you want to create a system model for the application under test in your team space.

Tasks	Go to...
Add a repository to the team space.	Adding a repository to a team space on page 173
Create a system model.	Creating a system model on page 175
Create components in a system model.	Creating components on page 177
Create child components to existing components in a system model.	Creating child components on page 178
Define the relationship between the components in a system model.	Creating linkages between components on page 180
Delete the relationship between the components in a system model.	Deleting linkages between components on page 182
Modify components in a system model.	Modifying components on page 184
Delete components in a system model.	Deleting components on page 186
Publish changes made to the system model.	Publishing changes to the system model on page 187
Associate resources with components in a system model.	Associating resources with components on page 187
View resources associated with components in a system model.	Viewing resources that are associated with components on page 189
Remove resources that are associated with components in a system model.	Removing resources that are associated with components on page 193
Publish changes of associating or dissociating resources in a project with components in the system model.	Publishing changes of associating or dissociating resources with components on page 195
View system model.	Viewing the system model on page 197

Tasks	Go to...
Delete system model.	Deleting a system model on page 197

Tasks for working with the team space repository

You can find the tasks that you can perform when you want to work with the team space repository.

Tasks	More information
Add a repository to the team space.	Adding a repository to a team space on page 173
Delete a team space repository.	Deleting a repository that is added to a team space on page 198

Adding a repository to a team space

You must add a repository to the team space when you want to create, modify, or store a system model.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Team Space Owner* or an *Architect* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you obtained valid credentials to access the Git repository that you want to add.

About this task

As a *Team Space Owner* or an *Architect*, you can add a repository to the team space. You can add a repository to the team space in any of the following methods:

- Add an existing repository that can contain test assets, which are created in the desktop products.
- Add a bare repository.



Attention: The repository that you want to add must not contain any projects or `.project` files under the root directory.


When you add a repository, the Git repository is cloned to the team space. While you add a repository, you must provide the necessary authentication credentials that are set for the Git repository. For example, if the authentication type is SSH, then you must provide the Git URL, a deploy key, and a passphrase.



Important: You can add only one repository to the team space. After you add a repository to the team space, the option to add another repository is not available. You must delete an existing repository before you can add a different repository.

1. Log in to Rational® Test Automation Server.
2. Open the team space to which you want a team space repository.

The **Active projects** page is displayed.

3. Click the **Settings** icon  from the left navigation pane, and then select **Manage team space**.

The **Details** page is displayed.

4. Click **Repository** on the left navigation pane.

The **Repository** page is displayed.

5. Click **Add repository**.

The **Add repository** dialog is displayed.

6. Enter the URL of the Git repository, which you want to add to the team space, in the **Git Repository** field.

The required fields are displayed based on the type of Git URL that you entered.

7. Enter the required credentials based on any of the following authentication methods configured in the repository.

To gain access to the repository, you must use any one of the authentication methods:

Authenticati- on method	Credentials required
SSH	<ul style="list-style-type: none"> ◦ Deploy key ◦ Passphrase
HTTPS	<ul style="list-style-type: none"> ◦ Username ◦ Password
HTTP	<ul style="list-style-type: none"> ◦ Username ◦ Password



Notes:



- You must have defined the authentication type and set the authentication credentials in the Git repository.
- If you use SSH to connect to your remote repository and Rational® Test Automation Server displays an `Auth Fail` exception while using the `deliver changes` option, you can resolve this exception error by regenerating your SSH keys by using the `-m PEM` option.

8. Click **Add**.

The Git repository is added to the team space on Rational® Test Automation Server.



Note: The time to clone the repository can take a few to several minutes and depends on the size of the repository.

What to do next

You can perform the following actions on the repository that you added:

- Update the authentication credentials if they are changed in the Git repository configuration. See [Updating the authentication credentials of the repository on page 635](#).
- Refresh a repository to fetch and synchronize changes from the remote repository. See [Refreshing repositories manually on page 636](#).
- Configure a webhook to notify the server if there is a push event in the remote repository. See [Creating webhooks on page 639](#).
- Add a system model to the repository. See [Creating a system model on page 175](#).
- Delete a repository that is added to the team space, if it is no longer required. See [Deleting a repository that is added to a team space on page 198](#).

Related information

[Managing repositories on page 632](#)

Creating a system model

Before you can create components in a system model, you must create an empty system model and publish the system model to the team space repository.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as an *Architect* in the team space, a *Project Owner* or a *Tester* in the project. See [Managing members and their roles in a team space on page 612](#) or [Managing access to server projects on page 627](#).
- Added a repository to the team space. See [Adding a repository to a team space on page 173](#).

1. Log in to Rational® Test Automation Server.
2. Open the team space to which a team space repository is added.

The **Active projects** page is displayed.

3. Select an option to create a system model based on your role in the team space or in a project in the team space:

- If you are a *Team space owner* or an *Architect*, go to [Step 4 on page 176](#).
- If you are not a *Team space owner* or an *Architect* but a project *Owner* or *Tester*, go to [Step 5 on page 176](#).

4. Perform the following steps, if you are a *Team space owner* or an *Architect*:

- a. Click **System Model** on the left navigation pane.

The **System Model** page is displayed.

- b. Click the **Add system model** icon .

An empty system model is created.

- c. Go to [Step 6 on page 176](#).

5. Perform the following steps, if you are not a *Team space owner* or an *Architect* but a *Project owner*:

- a. Open your project that is displayed under **My projects** on the **Active projects** page.

The **Overview** page is displayed.


- b. Click **System Model** on the left navigation pane.

The **System Model** page is displayed.

- c. Click the **Add system model** icon .

An empty system model is created.

- d. Go to [Step 6 on page 176](#).

6. Click the  icon, and then click the **Publish changes** option.



Note: If you do not want to commit the changes that you made to the system model, you can select the **Discard changes** option.

7. Enter a commit message in the **Description of change** field, and then click **Publish**.

The empty system model is committed and published to the team space repository. When the system model is published to the team space repository, other members of the team space can work with the same system model from any of their projects in the team space.

Results

You have created an empty system model and published it to the team space repository.

What to do next

You can perform any of the following tasks:

- Create components in the system model. See [Creating components on page 177](#).
- Create child components for the components that you added to the system model. See [Creating child components on page 178](#).

Creating components

After you create an empty system model in the team space, you can create components that represent components of the system under test.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as an *Architect* in the team space, a *Project Owner* or a *Tester* in the project. See [Managing members and their roles in a team space on page 612](#) or [Managing access to server projects on page 627](#).
- Added a repository to the team space. See [Adding a repository to a team space on page 173](#).
- Created and published the system model. See [Creating a system model on page 175](#).


1. Log in to Rational® Test Automation Server.
2. Open the team space to which a team space repository is added.

The **Active projects** page is displayed.

3. Click **System Model** on the left navigation pane.

The **System Model** page is displayed.




4. Perform the following steps to add a component to the system model:

- a. Click the **Add component** icon  in the **Components** row on the right pane.

The **Add a component** dialog is displayed.

- b. Enter a name for the component in the **Name** field.

- c. Select the type of component that you want to add from the following options displayed in the **Type** field.

-  **Service**
-  **Database**
-  **UI**

d. Click **Add**.

The component is displayed as a circular block on the system model pane with the icon and name of the component below the block. The added component is also displayed under the **Components** list on the right pane.

You can create components of other types or the same type in the system model by following the preceding steps.

Results

You have created components in the system model.

What to do next

You can perform any of the following tasks:

- Publish the changes that you made to the system model. See [Publishing changes to the system model on page 187](#).
- Create child components for the components that you added to the system model. See [Creating child components on page 178](#).
- Create linkages between components in the system model. See [Creating linkages between components on page 180](#).

Creating child components

You can create child components in the system model when you want to represent the sub-components of a component in a system under test.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as an *Architect* in the team space, a *Project Owner* or a *Tester* in the project. See [Managing members and their roles in a team space on page 612](#) or [Managing access to server projects on page 627](#).
- Created and published the system model. See [Creating a system model on page 175](#).
- Created components in the system model. See [Creating components on page 177](#).

About this task

You can create multiple components in your system model and multiple child components under any component in the system model. After you create a child for a component, the child is represented as a circular block within the circular component block in the system model.

1. Log in to Rational® Test Automation Server.
2. Open the team space to which a team space repository is added.


The **Active projects** page is displayed.

3. Click **System Model** on the left navigation pane.

The **System Model** page is displayed with the existing components.




4. Create child components by using any of the following options:
 - To create child components from the left pane on the **System Model** page, go to [Step 5 on page 179](#).
 - To create child components from the right pane on the **System Model** page, go to [Step 6 on page 179](#).
5. Perform the following steps to create child components from the left pane:
 - a. Click the component under which you want to create child components.

The details of the component are displayed in the right pane.

- b. Click the **Add child** icon  in the **Children** row on the right pane.

The **Add a child to...** dialog is displayed.

- c. Enter a name for the component in the **Name** field.
- d. Select the type of component that you want to add from the following options displayed in the **Type** field.


-  **Service**
-  **Database**
-  **UI**

- e. Click **Add**.

The child component is displayed as a circular block within the circular block of the parent component on the system model pane. The added child component is also displayed under the **Children** list under the component on the right pane.

You can create child components of other types or the same type for the same or other components in the system model by following the preceding steps.

6. Perform the following steps to create child components from the right pane of the stem model page:

- a. Click the component name listed under the **Components** in the right pane.
- b. Click the **Add child** icon  in the **Children** row on the right pane.
The **Add a child to...** dialog is displayed.
- c. Enter a name for the component in the **Name** field.
- d. Select the type of component that you want to add from the following options displayed in the **Type** field.

-  **Service**
-  **Database**
-  **UI**

- e. Click **Add**.

The child component is displayed as a circular block within the circular block of the parent component on the system model pane. The added child component is also displayed under the **Children** list under the component on the right pane.

You can create child components of other types or the same type for the same or other components in the system model by following the preceding steps.

Results

A component is created and it is associated as a child with the selected component and the name of the child is displayed under the **Children** section on the right pane.

What to do next

You can perform any of the following tasks:

- Publish the changes that you made to the system model. See [Publishing changes to the system model on page 187](#).
- Create linkages between components in the system model. See [Creating linkages between components on page 180](#).

Creating linkages between components

You can create linkages between components to display the relationship between them. The linkages can be between components or between child components of the same or different parent components.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as an *Architect* in the team space, a *Project Owner* or a *Tester* in the project. See [Managing members and their roles in a team space on page 612](#) or [Managing access to server projects on page 627](#).
- Created and published the system model. See [Creating a system model on page 175](#).
- Created components in the system model. See [Creating components on page 177](#).
- Created child components. See [Creating child components on page 178](#).

About this task

You can create multiple components in your system model. You can also create multiple child components under any component in the system model. You can specify the dependency among the components.

1. Log in to Rational® Test Automation Server.
2. Open the team space to which a team space repository is added.

The **Active projects** page is displayed.

3. Click **System Model** on the left navigation pane.


The **System Model** page is displayed.


4. Perform the following steps to create a relationship between the components in the system model:

- a. Click the component that you want.

The details of the component are displayed on the right pane.

- b. Perform the actions in the following table to create dependencies between the components:

Option	Action
Depends on	<p>Perform the following steps, if the component that you selected is dependent on another component:</p> <ol style="list-style-type: none"> i. Click the Add dependency icon  in the Depends on row on the right pane. <p>The Add dependency dialog is displayed.</p> <ol style="list-style-type: none"> ii. Select the component from the list of components displayed under Depends on. iii. Click Add. <p>The components linked are listed under the Depends on section on the right pane. The linkage between the first to the second component that you selected is shown with an arrow that starts from the first and ends in the second component.</p>

Option	Action
Is a dependency of	<p>Perform the following steps, if the component that you selected has a dependency on another component:</p> <ol style="list-style-type: none"> i. Click the Add dependent icon  in the Is a dependency of row on the right pane. <p>The Add dependent dialog is displayed.</p> <ol style="list-style-type: none"> ii. Select the component from the list of components displayed under the Is a dependency of section. iii. Click Add. <p>The components linked are listed under Is a dependency of section on the right pane. The linkage between the first to the second component that you selected is shown with an arrow that starts from the second and ends in the first component.</p>

Results

You have created linkages between components in the system model.

What to do next

You can publish the changes that you made to the system model. See [Publishing changes to the system model on page 187](#).

Deleting linkages between components

When you do not want to retain the linkages between components that you created in the system model, you can delete the linkages.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as an *Architect* in the team space, a *Project Owner* or a *Tester* in the project. See [Managing members and their roles in a team space on page 612](#) or [Managing access to server projects on page 627](#).
- Created and published the system model. See [Creating a system model on page 175](#).
- Created components in the system model. See [Creating components on page 177](#).
- Created child components. See [Creating child components on page 178](#).
- Created linkages between components in the system model. See [Creating linkages between components on page 180](#).

1. Log in to Rational® Test Automation Server.
2. Open the team space to which a team space repository is added.

The **Active projects** page is displayed.

3. Click **System Model** on the left navigation pane.



The **System Model** page is displayed.

4. Perform the following steps to delete a relationship between the components in the system model:

- a. Click the component that you want.

The details of the component are displayed on the right pane.

- b. Select from the following types of relationships that you want to delete:

When...	Action
The component depends on another component.	<p>Perform the following steps, if the component that you selected is dependent on another component:</p> <ol style="list-style-type: none"> i. Select the component from the list of components that are displayed under the Depends on section on the right pane. ii. Click the Delete selected icon  inline with the Depends on row. <p>The Delete relationship dialog is displayed with the details of the component and the dependent component.</p> <ol style="list-style-type: none"> iii. Click Delete. <p>The component linkage is removed from the system model pane.</p>
The component has a dependency on another component.	<p>Perform the following steps, if the component that you selected has a dependency on another component:</p> <ol style="list-style-type: none"> i. Select the component from the list of components that are displayed under the Is a dependency of section on the right pane. ii. Click the Delete selected icon  inline with the Is a dependency of row. <p>The Delete relationship dialog is displayed with the details of the component and the component dependency.</p> <ol style="list-style-type: none"> iii. Click Delete. <p>The component linkage is removed from the system model pane.</p>

Results

You have created linkages between components in the system model.

What to do next

You can publish the changes that you made to the system model. See [Publishing changes to the system model on page 187](#).

Modifying components

At any point in time, after you create a component, you can modify and update the details of the components such as the component type, and the relationships with other components.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as an *Architect* in the team space, a *Project Owner* or a *Tester* in the project. See [Managing members and their roles in a team space on page 612](#) or [Managing access to server projects on page 627](#).
- Created and published the system model. See [Creating a system model on page 175](#).
- Created components in the system model. See [Creating components on page 177](#).
- Created child components. See [Creating child components on page 178](#).

1. Log in to Rational® Test Automation Server.
2. Open the team space to which a team space repository is added.

The **Active projects** page is displayed.



3. Click **System Model** on the left navigation pane.





The **System Model** page is displayed.

4. Click the component that you want to edit.

The component details are displayed on the right pane.

5. Perform the actions for the task to modify components as listed in the following table:

Task	Action
Change the component type.	Perform the following steps: <ol style="list-style-type: none"> a. Identify the component that you want to modify from the components listed under Components on the right pane. b. Click the component checkbox. <p style="margin-left: 40px;">The Edit selected icon  is enabled.</p> c. Click the Edit icon .

Task	Action
	<p>The Edit component dialog box is displayed.</p> <p>d. Select the component type.</p> <p>e. Click Modify</p> <p>The type of the component is modified.</p>
Change the name of a component.	<p>Perform the following steps:</p> <p>a. Identify the component that you want to modify from the components listed under Components on the right pane.</p> <p>b. Click the component checkbox.</p> <p>The Edit selected icon  is enabled.</p> <p>c. Click the Edit icon .</p> <p>The Edit component dialog box is displayed.</p> <p>d. Edit the component name in the Name field.</p> <p>e. Click Modify.</p> <p>The name of the component is modified.</p>
Change the parent of a component.	<p>Perform the following steps:</p> <p>a. Identify the component that you want to modify from the components listed under Components on the right pane.</p> <p>b. Click the component checkbox.</p> <p>The Change parent icon  is enabled.</p> <p>c. Click the Change parent icon .</p> <p>The Change parent dialog box is displayed.</p> <p>d. Select another parent for the selected components from those that are listed under the Parent list.</p> <p>e. Click Change.</p> <p>The parent of the component is changed.</p>

Results

You have edited components and updated the details successfully.

What to do next

You can publish the changes that you made to the system model. See [Publishing changes to the system model on page 187](#).

Deleting components

When you do not require a component, you can delete the component from the system model.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as an *Architect* in the team space, a *Project Owner* or a *Tester* in the project. See [Managing members and their roles in a team space on page 612](#) or [Managing access to server projects on page 627](#).
- Created and published the system model. See [Creating a system model on page 175](#).
- Created components in the system model. See [Creating components on page 177](#).

1. Log in to Rational® Test Automation Server.
2. Open the team space to which a team space repository is added.

The **Active projects** page is displayed.

3. Click **System Model** on the left navigation pane.

The **System Model** page is displayed.

4. Click the component that you want to delete.

The component details are displayed on the right pane.

5. Click the **Delete component** icon  in the row of the component name.

The **Delete component** dialog is displayed.

6. Click **Delete**.

Results

You have deleted a component from the system model.

What to do next

You can perform any of the following tasks:

- Publish the changes that you made to the system model. See [Publishing changes to the system model on page 187](#).
- Create components in the system model. See [Creating components on page 177](#).

Publishing changes to the system model

When you create, delete, or modify components in a system model, the changes are saved locally and are not visible to the other project members or team space members. You must publish the system model changes to the team space repository to make your changes visible to all users of the team space.

Before you begin

You must have completed the following tasks:



- Ensured that you are assigned a role as an *Architect* in the team space, a *Project Owner* or a *Tester* in the project. See [Managing members and their roles in a team space on page 612](#) or [Managing access to server projects on page 627](#).
- Created and published the system model. See [Creating a system model on page 175](#).
- Created components in the system model. See [Creating components on page 177](#).
- Created child components. See [Creating child components on page 178](#).
- Created linkages between components in the system model. See [Creating linkages between components on page 180](#).

1. Log in to Rational® Test Automation Server.
2. Open the team space to which a team space repository is added.

The **Active projects** page is displayed.

3. Click **System Model** on the left navigation pane.

The **System Model** page is displayed.

4. View the number of unpublished changes made to the system model that is displayed along with the  icon.
5. Click the  icon, and then click the **Publish changes** option.



Note: If you do not want to commit the changes that you made to the system model, you can select the **Discard changes** option.

6. Enter a commit message in the **Description of change** field, and then click **Publish**.

Results

You have published the changes that you made in the system model to the team space repository. Other members of the team space can view and use the system model.

Associating resources with components

After creating components in the system model, you can navigate to a project in the team space to associate the Suites, tests, or virtual service resources in the project repositories with components in the system model.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created and published the system model. See [Creating a system model on page 175](#).
- Created components in the system model. See [Creating components on page 177](#).

1. Log in to Rational® Test Automation Server.
2. Open the team space to which a team space repository is added.

The **Active projects** page is displayed.

3. Click **Active projects > My projects > *project_name*** to open the project that contains the test assets.

The **Overview** page of the project is displayed.

4. Click **System Model** on the left navigation pane.

The **System Model** page is displayed.

5. Click the component that you want to associate with the Suites, Tests, or virtual service resources.

The component details are displayed in the right pane.

6. Click the **Associate** tab, and then perform the following steps:

- a. Click **Add**.

- b. Select the type of resource that you want to associate with the component from the following options:

- Suites and Tests
- Virtual services



Note: You can select either of the options or both the options.

The **Edit branch** dialog is displayed.

- c. Select your action that depends on the existence of an *Edit* branch from the following options:

- Select an existing *Edit* branch that you created previously from the list of branches listed in the **Select branch** field, and then click **Save**.
- Click **Create new branch**, enter a name for the branch, and then click **Save**, if you want to create a new branch.

The **Associate Assets** dialog is displayed. The resources that are contained in the branch of the project repository are listed. The name of the component and the branch are also displayed.

The **Suites and Tests** option is selected and the assets are listed if you selected the **Suites and Tests** option.


The **Virtual services** option is selected and the virtual service resources are listed if you selected the **Virtual services** option.

If you selected both the types of resources, then assets and virtual services in the branch are listed.

- d. Identify the assets or virtual resources that you want to associate with the component in the system model by performing any of the following actions:
 - Search for the resource by entering the asset name in the **Search** field.
 - Find the assets by scrolling through the list of assets.
- e. Select the resources, and then click **Add**.

The selected resources are associated with the component.

You can view the associated resources that are displayed in the **Associate** tab and the **Overview** tab. The test assets are listed under the **Suites and Tests** section and the virtual service resources are listed under the **Virtual services** section.

If you want to associate other resources to the same component, you can add them by clicking the **Add association** icon  in the resource row on the right pane. You can then select the resource and add the resource to the selected component.

Results

You have associated Suites, tests, or virtual service resources with a component in the system model.

What to do next

You can perform any of the following tasks:

- Publish the changes made in the system model for associating resources with components. See [Publishing changes to the system model on page 187](#).

Viewing resources that are associated with components

After you associate Suites, tests, or virtual services resources with components in the system model, you can view the associated resources from the **System Model** page. You can opt to run an instance of the associated resource or stop a running instance of the associated resource.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created and published the system model. See [Creating a system model on page 175](#).
- Created components in the system model. See [Creating components on page 177](#).
- Associated resources in a project with components in the system model. See [Associating resources with components on page 187](#).

1. Log in to Rational® Test Automation Server.
2. Open the team space to which a team space repository is added.

The **Active projects** page is displayed.

3. Click **Active projects > My projects > project_name** to open the project that contains the test assets.

The **Overview** page of the project is displayed.

4. Click **System Model** on the left navigation pane.

The **System Model** page is displayed.




5. Click the component that you associated with the resources in the project repository.





The resources associated with the component are displayed under the resource type in the **Associate** tab.

6. Click the **Overview** tab, if it is not already displayed.






The resources that are associated with the component are displayed under the **Suites and Tests** or **Virtual Services** section.





7. View and perform the actions for the resource type that you want from the following options:
 - For **Suites and Tests** resources, go to [8 on page 190](#).
 - For **Virtual Services** resources, go to [9 on page 191](#).
8. Identify the test resource from those that are listed under the **Suites and Tests** section, and perform the action that you want as described in the following table:



Task	Description	Action
Collapse icon 	You can hide the resource sections by clicking the Collapse icon  .  Note: When the section is hidden, you can view the number that is dis-	Click the icon to toggle the display to show or hide the resource list in the Suites and Tests section.

Task	Description	Action
	 played of the resources that are associated.	
Type	You can view the type of Suite or test.	Hover over the Suites and Tests icon under the Type column to view the type of Suite or test.
Name	You can view the name of the associated Suite or test and the path in the repository.	There is no action to perform.
Status	You can view the details of the test runs.	Click each of the squares to view the details of the specific test run. For example, if the status of a test run is represented as shown in the image  , then it indicates that the Suite or test has run only two times. After you click any colored square, the Results page is displayed.
Action	You can start the execution of the Suite or test from the Execution page when you click the Show in execution page icon  .	Click the Show in execution page icon  to view the Suite or test on the Execution page. The Suite or test is displayed on the Execution page.

9. Identify the virtual service resource from those that are listed under the **Virtual Services** section, and perform the action that you want as described in the following table:

Field	Description	Action
Collapse icon 	You can hide the resource sections by clicking the Collapse icon  .	Click the icon to toggle the display to show or hide the resource list in the Virtual Services section.
	 Note: When the section is hidden, you can view the number that is displayed of the resources that are associated.	
Show all icon 	You can opt to view the virtual services that are associated with the component either on the Resources or Instances page.	Perform the following steps: <ul style="list-style-type: none"> a. Click the Show all icon . b. Select any of the following actions:

Field	Description	Action
	You can start an instance of the virtual service from the Resources page while you can stop a running instance from the Instances page.	<ul style="list-style-type: none"> ▪ To start an instance of the virtual service: <ol style="list-style-type: none"> i. Select Show all in resources page to go to the Resources page. ii. Click the Execute icon  in the row of the virtual service. iii. Configure the settings for the run of the virtual service, if you want to change any of the settings, else click Execute. <p>The virtual service starts to run.</p> ▪ To stop a running instance of the virtual service: <ol style="list-style-type: none"> i. Select Show all in instances page to go to the Instances page. ii. Identify the instance that you want to stop and click the Stop icon  in the Actions column of the selected virtual service instance. iii. Click Ok in the Stop virtual service instance dialog that is displayed. <p>The running virtual service instance is stopped.</p>
Name	You can view the name of the associated virtual service and the path in the repository.	There is no action to perform.
Instances	You can view the state of the running instance of the virtual service resource. The state is displayed as <code>Running</code> only if the instance is running.	There is no action to perform.
Action	You can go to the Resources page to view the virtual service resource when you click the Show resource icon  .	<p>Click the Show resource icon  to view the virtual service on the Resources page.</p> <p>The virtual service resource is displayed on the Resources page.</p>

Field	Description	Action
	You can start an instance of the virtual service from the Resources page.	
Expand icon 	<p>You can expand the virtual service card only if the virtual service is a running instance.</p> <p>In the expanded card, you can view the details such as the Environment, the total number of requests received by the virtual service, and the relative time from when the virtual service had started.</p> <p>You can stop a running instance of the virtual service from the Instances page.</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Expand the virtual service card for a running instance. <p>You can view the details of the instance.</p> <ol style="list-style-type: none"> Click the Show instance icon  to go to the Instances page to view the running instance of the virtual service. <p>The running instance of the virtual service is displayed on the Instances page.</p>

Results

You have viewed the resources that are associated with a component from the **System Model** page.

Removing resources that are associated with components

When you do not want to retain the resource that you associated with a component, you can delete the resource association from the system model.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created and published the system model. See [Creating a system model on page 175](#).
- Created components in the system model. See [Creating components on page 177](#).
- Associated resources in a project with components in the system model. See [Associating resources with components on page 187](#).

1. Log in to Rational® Test Automation Server.
2. Open the team space to which a team space repository is added.

The **Active projects** page is displayed.

3. Click **Active projects > My projects > *project_name*** to open the project that contains the test assets.

The **Overview** page of the project is displayed.

4. Click **System Model** on the left navigation pane.

The **System Model** page is displayed.

5. Click the component that you associated with the resources in the project repository.

The resources associated with the component are displayed under the resource type in the **Associate** tab.

6. Select the resource that you want to remove from the component by clicking the checkbox of the resource.

You can select any or all resources for removal.

7. Click the **Delete selected** icon  in the row of the resource type.

The **Disassociate...** dialog is displayed for the type of resource that you selected. For example, the **Disassociate Suites and Tests** dialog is displayed when you selected the test assets.

8. Click **Delete**.

The **Edit branch** dialog is displayed.

9. Select your action that depends on the existence of an *Edit* branch from the following options:

- Select an existing *Edit* branch that you created previously from the list of branches listed in the **Select branch** field, and then click **Save**.
- Click **Create new branch**, enter a name for the branch, and then click **Save**, if you want to create a new branch.

The dialog that is displayed depends on the type of resource you selected and can be any of the following types:

- The **Dissociate suites and tests** dialog is displayed, if you selected Suites or tests as the resources to dissociate.
- The **Dissociate virtual services** dialog is displayed, if you selected the virtual services as the resources to dissociate.

10. Click **Delete**.

Results

You have removed resources that were associated with components in the system model.

What to do next

You must publish the dissociation changes in the system model to the project repository if you want the changes to be reflected in the system model for all members of the project. See [Publishing changes of associating or dissociating resources with components on page 195](#).

Publishing changes of associating or dissociating resources with components

After you associate or dissociate resources in a project with a component in the system model, you must publish the changes made to the branch in the project repository so that the association changes are visible to the other members of the project.

Before you begin

You must have completed the following tasks:

- Created and published the system model. See [Creating a system model on page 175](#).
- Created components in the system model. See [Creating components on page 177](#).
- Associated resources in a project with components in the system model. See [Associating resources with components on page 187](#).
- Dissociated resources from components in the system model. See [Removing resources that are associated with components on page 193](#).

1. Log in to Rational® Test Automation Server.
2. Open the team space to which a team space repository is added.



The **Active projects** page is displayed.

3. Click **Active projects > My projects > project_name** to open the project that contains the test assets.

The **Overview** page of the project is displayed.

4. Click **System Model** on the left navigation pane.

The **System Model** page is displayed.

5. View the number of uncommitted changes made to the system model that is displayed along with the  icon.
6. Click the  icon.

You can view the options for unpublished changes made to the components and associations in the system model. The options available for association changes are as follows:

- Review changes
 - Discard changes
 - Publish changes
7. Perform any of the following actions to publish the association or dissociation changes:
 - Review the unpublished changes before you publish them. Go to [Step 8 on page 195](#).
 - Publish the changes. Go to [Step 9 on page 196](#).
 8. Perform the following steps to review the unpublished changes:

- a. Select the **Review changes** option.

The **Changes** page is displayed. You can find the following information about the changes that you made to the system model when you associated resources with components or deleted the association:

- The *Edit* branch that you selected to contain the changes is displayed in the **Branch** field. You can select the *Edit* branch that you used before associating resources to components, if it is not already displayed as selected.
- The projects that contained the resources, which you associated with components in the system model are listed under the **Asset name** column.
- The type of change is listed under the **Changes** column.
- The name of the member who made the change is displayed under the **Last changed by** column.
- The relative time when the change was made is displayed under the **Last updated** column.
- The number of changes that the *Edit* branch is ahead of from the project repository branch is indicated by the **commits ahead** option. You can view the details by clicking the **commits ahead** option and in the dialog that displays, you can discard the pending commits by clicking the **Reset** option.
- The number of changes that the *Edit* branch is behind from the project repository branch is indicated by the **commits behind** option. The *Edit* branch can be behind the project repository branch if another member has published or committed other changes in the system model to the same project repository branch. You can view the details by clicking the **commits behind** option. You can merge all the changes to your *Edit* branch so that the *Edit* branch is updated with all the changes made in the project repository branch.

- b. Click **Publish changes**.

- c. Enter a commit message in the **Description of change** field, and then click **Publish**.



Note: If you do not want to commit the changes that you made to the system model, you can select the **Discard changes** option.

9. Perform the following steps to publish the unpublished changes:

- a. Click the **Publish changes** option.

- b. Enter a commit message in the **Description of change** field, and then click **Publish**.



Note: If you do not want to commit the changes that you made to the system model, you can select the **Discard changes** option.

Results

You have published the association changes that you made in the system model to the repository in the project. Other members of the team space can view the association changes and use the system model.

Viewing the system model

You can view an existing system model that is in a team space if you are a member of the team space.

Before you begin

You must be a licensed user of Rational® Test Automation Server and a member of a team space.

1. Log in to Rational® Test Automation Server.
2. Open the team space to which a team space repository is added.

The **Active projects** page is displayed.

3. Click **System Model** on the left navigation pane.

The **System Model** page is displayed.

Results

You viewed an existing system model in the team space from the **System Model** page.

Deleting a system model

When you want to remove an existing system model to create a new system model, you must delete the existing system model because only one system model is supported in a team space.

Before you begin


You must have ensured that a system model exists in the team space.

1. Log in to Rational® Test Automation Server.
2. Open the team space to which a team space repository is added.

The **Active projects** page is displayed.

3. Click **System Model** on the left navigation pane.

The **System Model** page is displayed.

4. Click the **Open action menu** icon  in the right pane.
5. Click the **Delete** icon .

The **Delete System Model** dialog is displayed.

6. Click **Delete**.

Results

You deleted the system model that existed in the team space.

What to do next

You must publish the changes to the team space repository. See [Publishing changes to the system model on page 187](#).

Deleting a repository that is added to a team space

You can delete a repository that you added to the team space when you do not want to use a team space repository.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Team Space Owner* or an *Architect* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Added a repository to the team space. See [Adding a repository to a team space on page 173](#).

1. Log in to Rational® Test Automation Server.
2. Open the team space to which a team space repository is added.


The **Active projects** page is displayed.

3. Click the **Settings** icon  from the left navigation pane, and then select **Manage team space**.

The **Details** page is displayed.

4. Click **Repository** on the left navigation pane.

The **Repository** page is displayed.

5. Click the **Open action menu** icon .
6. Click **Delete** in the action menu list.

The **Delete repository** dialog is displayed.

7. Select the **I understand...** checkbox, and then click **Delete**.

Results

You have deleted the team space repository.



Restriction: When you delete the team space repository, the system model if it exists is also deleted, and you or other members of the team space cannot add a system model.

Prerequisites to running tests

Before you can run a test asset in a project on Rational® Test Automation Server, you must read the information about the different tests assets. You can add a remote static agent or Docker host to the project and then select it as an alternate location to run the test assets.

You can find the following information about the prerequisite tasks:

- [Test run considerations for AFT Suites or Functional tests on page 199](#)
- [Test run considerations for API Suites or API tests on page 200](#)
- [Test run considerations for schedules on page 203](#)
- [Considerations for using Jaeger traces in reports on page 205](#)
- [Test run considerations for JMeter tests on page 205](#)
- [Test run considerations for JUnit tests on page 207](#)
- [Test run considerations for Postman tests on page 208](#)
- [Management of performance agents on page 210](#)
 - [Test run considerations for running tests on remote performance agents on page 211](#)
- [Management of Docker hosts on page 218](#)
 - [Test run considerations for running tests on remote Docker hosts on page 218](#)
- [Management of remote Kubernetes clusters on page 234](#)
 - [Considerations for using a remote Kubernetes cluster on page 235](#)

Related information

[Test run configurations on page 249](#)

Test run considerations for AFT Suites or Functional tests

Before you configure an AFT Suite or a Functional test run, you must first read the considerations that you need to take into account.

When you want to run an AFT Suite or Functional test in a project on Rational® Test Automation Server, you can check if the AFT Suite or Functional test meets any of the following conditions:

If...	Then...
The AFT Suite or Functional test uses the settings configured in an AFT XML file for a remote agent	<p>You must ensure that you have configured the location element in the AFT XML to point to a remote agent.</p> <p>For more information about setting the location element, refer to Using an XML file to run multiple Web UI tests and compound tests simultaneously.</p>
You want to run an AFT Suite or Functional test on a remote agent	<p>You must run the test on the remote agent before you commit the test asset to the remote repository.</p> <p>When you run the test on the remote agent, the following events occur:</p>

If...	Then...
	<ul style="list-style-type: none"> • The agent is added as the location in the AFT XML file, on which the test is to be run. • The agent is displayed as the agent on which the test is to be run, under the Host column within the Location tab in the Execute test asset dialog.
You commit the test asset to the remote repository without running the test on the remote agent	The agent is not displayed under the Host column within the Location tab in the Execute test asset dialog.

Before you configure the run for an AFT Suite or a Functional test, you must complete the following tasks, if they are applicable:

- Add the remote agents, on which the test is to be run, to the project on Rational® Test Automation Server. See [Adding a performance agent to a project on page 213](#).
- After you add the agent, you can select the agent configured in the AFT XML file as the location for the test run or select the remote agents that you add to the project, when you configure a run for an AFT Suite.

The remote agents are shown as the agents available for selection under the **Override** column within the **Location** tab in the **Execute test asset** dialog.

You can then select the remote agent as the location to run the AFT Suite or Functional test when you are configuring the test run. Alternatively, you can enter the argument `-swaplocation <configured_agent_location>:<overriding_agent_location>` in the **Program Arguments** field in the **Advanced Setting** dialog when you are configuring the test run.

For running an AFT Suite, see [Configuring an AFT Suite run on page 255](#).

Related information

[Test run configurations on page 249](#)

Test run considerations for API Suites or API tests

Before you configure an API Suite or API test run, you must first read the considerations that you need to take into account.

You can find the following information about API Suites:

- [Important information about API Suites or tests on page 201](#)
- [Important information about report configuration in API Suites or tests on page 203](#)

Important information about API Suites or tests

When you want to run an API Suite or test from a project on Rational® Test Automation Server and the test asset in Rational® Integration Tester meets any of the following conditions:

- The Suite or test refers to local stubs that use a transport other than HTTP or MQ.
- The Suite or test refers to local stubs that use the HTTP transport.
- The transport used in the tests or Suite is configured with the host name set to *localhost* for the HTTP/TCP proxy.
- The Suite or test has tests that use a transport and the transport requires third-party application `JAR` files for a successful run.

You must then refer to the following table for the next steps:

If...	Then...
<p>The test asset refers to local stubs that use a transport other than HTTP or MQ.</p>	<p>You must perform any of the following actions before you commit the test asset to the remote repository:</p> <ul style="list-style-type: none"> • Remove the reference to the local stub from the test asset. • Publish the stubs to Rational® Test Control Panel. <p>To publish and edit stubs, see Publishing stubs.</p>
<p>The tests refer to local stubs that use the HTTP transport.</p>	<p>You must perform the following actions before you commit the test asset to the remote repository:</p> <ol style="list-style-type: none"> 1. Create a test or test Suite that contains the stub and tests that run on the stub. 2. Configure the physical HTTP transport with the following parameters in the Client tab in the Physical View: <ul style="list-style-type: none"> ◦ Set the <code>Host name or IP address of the HTTP Proxy server</code> in the Proxy Host field. ◦ Set <code>3128</code> as the proxy server port in the Proxy Port field. 3. Set the reference to the stub for the tests in the test suite scenario from the Scenario Editor dialog from the Test Factory view by selecting the <i>Live system</i> option in the Satisfied by column to

If...	Then...
	<p>run the stubs on the HTTP proxy registered with Rational® Test Automation Server.</p> <p>4. Save the project.</p>
<p>The transport used in the Suite or tests is configured with the host name set to <i>localhost</i> for the HTTP/TCP proxy.</p>	<p>You must replace the host name with <i>fully-qualified-domain-name</i> or <i>IPAddress</i> of the proxy host in the tests before you commit the test asset to the remote repository.</p>
<p>You plan to run the API Suite or test on the computer where Rational® Test Automation Server is installed and the API Suite or test that use a transport from any of the following third-party applications:</p> <ul style="list-style-type: none"> • Camel • CentraSite • CICS • Coherence • Database • IMS • Integra • JMS • SAP RFC • Software AG Universal Messaging • TIBCO EMS • TIBCO Rendezvous • TIBCO SmartSockets • WebSphere Application Server Service Integration Bus (SiBus) • WebLogic • Software AG webMethods • WebSphere MQ 	<p>You must complete the following tasks:</p> <ol style="list-style-type: none"> 1. Identify the location of the third-party application JAR files. You can use Library Manager to know the location where the third-party application JAR files are saved. 2. Copy the third-party application JAR files to the computer where Rational® Test Automation Server is installed. <p>See Copying third-party application Jars to Kubernetes on page 118.</p>
<p>You plan to run the API Suite or test on a remote Docker host and the API Suite or test contains tests that use</p>	<p>You must complete the following tasks:</p>

If...	Then...
<p>a transport from any of the following third-party applications:</p> <ul style="list-style-type: none"> • Camel • CentraSite • CICS • Coherence • Database • IMS • Integra • JMS • SAP RFC • Software AG Universal Messaging • TIBCO EMS • TIBCO Rendezvous • TIBCO SmartSockets • WebSphere Application Server Service Integration Bus (SiBus) • WebLogic • Software AG webMethods • WebSphere MQ 	<ol style="list-style-type: none"> 1. Identify the location of the third-party application JAR files. You can use Library Manager to know the location where the third-party application JAR files are saved. 2. Copy the third-party application JAR files to the computer where the remote Docker host is installed. <p>See Copying third-party application Jars to a remote Docker host on page 223.</p>

Important information about report configuration in API Suites or tests

You must configure test Suites or tests in Rational® Integration Tester to use a results database so that the details of the test results can be captured and displayed in the Rational® Test Automation Server API results reports. See [Configuring the project results database](#).

If your server projects use either a Microsoft SQL Server, MySQL, or both as a project results database, you must copy the Microsoft SQL Server and MySQL database JAR files to the JDBC folder in the path `/data/JDBC` on IBM® Rational® Test Automation Server. See [Copying third-party application Jars to Kubernetes on page 118](#).

The database JAR files that you need are based on the version of Microsoft SQL Server or MSQL that you use with Rational® Integration Tester. See [Adding Microsoft SQL Server and MySQL drivers](#).

To run an API Suite, see [Configuring an API Suite or an API test run on page 267](#).

Test run considerations for schedules

Before you configure a Rate Schedule or VU Schedule run, you must first read the considerations that you need to take into account.

Important information about JMeter tests in schedules

When you want to run JMeter tests as part of the VU Schedule or Rate Schedule on Rational® Test Automation Server, you must have completed the following tasks:

- Installed the IBM® Rational® Performance Tester Agent on a computer.
- Installed the JMeter application on the computer where you have installed the IBM® Rational® Performance Tester Agent.
- Set the environment variable **JMETER_HOME** that points to the JMeter installation directory.

Important information about Resource Monitoring sources in schedules

When you want to run schedules on Rational® Test Automation Server, you must have completed the following tasks in Rational® Performance Tester:

- Created a performance schedule of **Rate** or **VU** type.
- Added Resource Monitoring sources, or the labels of Resource Monitoring sources to a schedule. Refer to [Adding resource monitoring sources to a performance schedule by using labels](#) .


Before you configure a run of a Schedule, you must complete the following tasks in Rational® Test Automation Server:

- Add Resource Monitoring sources to your project in Rational® Test Automation Server. See [Monitoring an NGINX server on page 490](#).
- Enable Rational® Test Automation Server as Resource Monitoring service only if you want to replace Resource Monitoring labels set in Rational® Performance Tester with the labels that you created in Rational® Test Automation Server. See [Controlling resource monitoring sources in a schedule on page 521](#).
- Perform the following steps when you want to override the Resource Monitoring labels in the test asset with the Resource Monitoring labels that you create in Rational® Test Automation Server:
 1. Click the **Resource Monitoring** tab.



Note:

The **Resource Monitoring** tab displays only if you enabled the **Resource Monitoring from Service** option in Rational® Performance Tester when you created the test asset.

2. Click , and press the `Ctrl + Space bar` keys, or enter the initial letter of a label to select a label in the list.

**Note:**

You can select or add labels only if you added the labels in the **Resource Monitoring Sources** page in Rational® Test Automation Server.

For running a Rate Schedule or VU Schedule, see [Configuring a run of a Rate Schedule or VU Schedule on page 371](#).

Related information

[Test run configurations on page 249](#)

Considerations for using Jaeger traces in reports

Before you can use Jaeger to report the test results in the supported tests, you must read the considerations that you must take into account.

When you want to enable Suites, Tests, or Schedules that are run from Rational® Test Automation Server to use Jaeger traces for reporting the test results, you must have completed the following tasks:

- Installed Jaeger at the time of installation of the server software. See [Prerequisites for installing the server software on IBM Red Hat OpenShift on page 30](#) or [Prerequisites to install the server software on Ubuntu on page 47](#).
- Verified that Jaeger is enabled and working.

After you install and verify that Jaeger is enabled, and when you want Jaeger to report the test results for the tests that you configure for a run, you must perform the following steps:

1. Click **Advanced**.
2. Enter `-history jaeger` in the **Program Arguments** field.

Related information

[Test run configurations on page 249](#)

Test run considerations for JMeter tests

Before you can run JMeter tests on IBM® Rational® Test Automation Server, you must read the considerations that you must take into account and complete the tasks indicated.

When you want to run JMeter tests that you create in JMeter, you must have completed the following tasks:

- You must ensure that the JMeter extension on IBM® Rational® Test Automation Server is enabled at the time of installation of the server software or is running before you add the repository that contains JMeter tests to the server project. See [Prerequisites for installing the server software on IBM Red Hat OpenShift on page 30](#) or [Prerequisites to install the server software on Ubuntu on page 47](#).
- You must have committed the following JMeter assets or resources as a JMeter project to the remote repository:



Note: You must have created a project directory and added the associated tests or assets in sub directories, that might be required for the JMeter tests to run successfully.

- The `jmeterRoot.jprj` project file in the project directory.
- The JMeter tests in a sub directory with a name you provide within the project directory.
- Optionally, the Java Key Store that is required at test run time in the `certs` sub directory.
- Optionally, the JMeter properties files that are required at test run time in the `properties` sub directory.
- Optionally, the library jar files that are required at test run time in the `lib` sub directory.
- Optionally, the test dependencies that are required at test run time in the `deps` sub directory.

For example, the project directory can be *myproject* that contains the `jmeterRoot.jprj` project file. In the project directory, you can create the following sub directories:

- The `mytests` sub directory that contains the JMeter tests.
- Optionally, the `certs` sub directory that contains the Java Key Store.

For example, the path to the key store can be specified in the `system.properties` file as follows:

```
javax.net.ssl.trustStore=$(deploy.home)/certs/simpleStore.jks
```

Where `<simpleStore.jks>` is the Java Key Store and the `<deploy.home>` is the keyword to access the key store at the test run time.

- Optionally, the `properties` sub directory that contains properties files such as `system.properties`, `jmeter.properties`, `reportGenerator.properties`, or `user.properties`.
- Optionally, the `lib` sub directory that contains the **lib** and **lib/ext** content to be used at test run time. For example, the **amq client** and the **plugins** required to parse the **jmx** that contains **amqp** calls.
- Optionally, the `deps` sub directory that contains the test dependencies.

The resulting project directory can be as follows:

```
myproject (that contains the jmeterRoot.jprj)
myproject/mytests
myproject/certs
myproject/properties
myproject/lib
myproject/deps
```

- You must ensure that you use a relative path when JMeter tests refer to other test assets. For example, if the JMeter test that is in the `myproject/mytests` folder refers to a `mycsv.csv` file that is in the `myproject/deps` folder, then the `mycsv.csv` is referred as follows: `../deps/mycsv.csv` in the JMeter test.
- You can create the JMeter project file named as `jmeterRoot.jprj`.

The project file is a java properties file. The JMeter project file is required so that IBM® Rational® Test Automation Server can identify the asset as a JMeter project. You can specify additional information in the project file. You can specify the names of the sub directories created and through keywords you can specify the path in the properties files whether they are relative to the root directory or to the tests sub directory.

For example, if you have created the following sub directories in the project directory called *myproject* that contains the `jmeterRoot.jprj` project file:

- `myproject/mytests`
- `myproject/properties`

If the JMeter tests are contained in the following path: `myproject/mytests/testplan/test1.jmx`, and some dependencies that are also required to be deployed at test run time exists in the following path: `myproject/mytests/testplan/localdir`. You must specify these file paths in the project file as follows:

```
testDir.localtest.sub.dir=localdir
aSecondRootDir.sub.dir=dir2
athirdRootDir.sub.dir=dir3
```



Restriction: You cannot run the test assets on a remote Docker host.

Related reference

[Troubleshooting issues on page 663](#)

Related information

[Configuring a run of a JMeter test on page 344](#)

Test run considerations for JUnit tests

Before you can run JUnit tests on IBM® Rational® Test Automation Server, you must read the considerations that you must take into account and complete the tasks indicated.

When you want to run JUnit tests, you must have completed the following tasks:

- You must ensure that the JUnit extension on IBM® Rational® Test Automation Server is enabled at the time of installation of the server software or is running before you add the repository that contains the JUnit tests to the server project. See [Prerequisites for installing the server software on IBM Red Hat OpenShift on page 30](#) or [Prerequisites to install the server software on Ubuntu on page 47](#).

- You must have created the JUnit tests that use JDK 8, as part of a Maven V3.6.3 project. The Maven project must contain the Maven Surefire plug-in, which is required to run the JUnit tests contained in the Maven project. The Maven project as a `pom.xml` must be committed to the remote repository.



Note: If you create the JUnit tests on other versions of JDK, you must copy the JDK version to the `userlibs` pod under the `data/junit-ext/jdks` directory. See [Copying third-party application Jars to Kubernetes on page 118](#).



Restriction: You cannot run the test assets on a remote Docker host.

Related reference

[Troubleshooting issues on page 663](#)

Related information

[Configuring a run of a JUnit test on page 353](#)

Test run considerations for Postman tests

Before you can run Postman tests on IBM® Rational® Test Automation Server, you must read the considerations that you must take into account and complete the tasks indicated.

When you want to run Postman tests that you create in Postman, you must have completed the following tasks:

- You must ensure that the Postman extension on IBM® Rational® Test Automation Server is enabled at the time of installation of the server software or is running before you add the repository that contains Postman collections to the server project. See [Prerequisites for installing the server software on IBM Red Hat OpenShift on page 30](#) or [Prerequisites to install the server software on Ubuntu on page 47](#).



Restriction: If you add the repository with Postman collections to a server project before you enable the Postman microservice, the Postman collections might not be displayed in the **Execution** page for you.

- You must have exported the following resources from Postman and saved the resources on your computer:
 - Collections and their associated variables and environments from Postman in which all collections are exported as a single file.



Note: You can export as many collections as you want.

For more information, refer to [Exporting a collection from Postman](#) and [Exporting data dumps](#).

- Environments configured in Postman, if you are exporting only the environments as separate JSON files.

For more information, refer to [Exporting environments from Postman](#).

- You must have committed the following Postman assets or resources to the remote repository:
 - Collections and their associated variables and environments that you exported. You can export all resources that are in your workspace in Postman. You can also export the collections along with the variables as a single file and the environments separately as another file.

For example, you can save the exported file as `MyCollectionsVarEnv.json` that contains the collections, variables, and environments or you can export only the collections and variables as `MyCollectionsVar.json`.

- Environments that you exported, if you are exporting only the environments as separate JSON files. For example, you can save the exported file as `MyEnv.json`.
- An empty text file with the `.postman` extension that is contained in the same directory as the other Postman resources, so that the contents in the directory are attributed as Postman resources. For example, you can create `MyProject.postman`.



Note: The `.postman` file enables IBM® Rational® Test Automation Server to identify the test resources as Postman resources.

For example, if you have saved the Postman resources that you downloaded from Postman as follows:

- `MyCollectionsVarEnv.json` or `MyCollectionsVar.json` that contains the Postman collections.
- `MyEnv.json` that contains the environments associated with the Postman collection.

The project directory can be as follows:

```
/Project
/Project/MyProject.postman
/Project/MyCollectionsVarEnv.json
```

Alternatively, the directory can be as follows:

```
/Project
/Project/MyProject.postman
/Project/MyCollectionsVar.json
/Project/MyEnv.json
```



Restriction: You cannot run the test assets on a remote Docker host.

Related reference

[Troubleshooting issues on page 663](#)

Related information

[Configuring a run of a Postman test on page 362](#)

Management of agents

You can find information about managing performance agents that you install on remote computers and then configure the remote agents to register with IBM® Rational® Test Automation Server. After the agents are registered, you can add them to your projects and then select them as locations to run tests.

When you plan to run any of the tests from a project on Rational® Test Automation Server on a remote performance agent, see [Management of performance agents on page 210](#).

Management of performance agents

You can manage the performance agents that you want to use as locations to run tests from IBM® Rational® Test Automation Server. When you install the agents by using your offline user token, the agents are registered with the team space in IBM® Rational® Test Automation Server. You can view the registered agents before you add them to your projects. You can then select them as locations to run tests.

When you plan to run any of the tests on a remote performance agent from a project in Rational® Test Automation Server, you must have completed the following tasks:

- Installed the performance agent of the same version as the version of Rational® Test Automation Server. Refer to the IBM® Rational® Performance Tester documentation for installing the performance agent at [Installing Rational Performance Tester agent](#).
- Configured the performance agent with your offline user token so that the performance agent can register with Rational® Test Automation Server.



Note: When you configure the performance agent by using your offline user token, Rational® Test Automation Server considers you as the owner of the performance agent.

- Verified that you have write permissions to the `majordomo.config` file so that certain transactions can be written by Rational® Test Automation Server to the `majordomo.config` file.



Note: You can verify if you have the *Read* and *Write* permissions by checking the properties of the `majordomo.config` file.

- Viewed the performance agents that are registered with Rational® Test Automation Server. See [Viewing performance agents that are registered with Rational Test Automation Server on page 212](#).
- Added the remote performance agents that are registered with Rational® Test Automation Server to your project on Rational® Test Automation Server. See [Adding a performance agent to a project on page 213](#).
- After you add the remote performance agents to your project, Rational® Test Automation Server determines the capabilities that are provided by the remote performance agent and displays the details and capabilities

of the performance agent on the **Agents and Intercepts** page. You can also define the capabilities that you want to add, and then view, edit, or delete the capabilities that you defined for the performance agents in your project. See [Working with the capabilities of a performance agent on page 215](#).

Test run considerations for running tests on remote performance agents

Before you configure a test to run on a remote performance agent from IBM® Rational® Test Automation Server, you must read the considerations that you must take into account.

Supported tests

You can run the following tests that are supported to be run on a remote performance agent:

- AFT Suite
- Compound Tests that contains performance tests
- Compound Tests that contains Web UI tests
- Compound Tests that contains traditional HTML tests
- Rate Schedule
- VU Schedule

Prerequisite tasks

Before you can run any of the supported tests on a remote performance agent from a project on Rational® Test Automation Server, you must have completed the following tasks:

- Installed the IBM® Rational® Performance Tester agent on the remote computer. See [Management of performance agents on page 210](#).
- Configured the agent with your offline user token so that the agent can connect to Rational® Test Automation Server.
- Added the remote agents to your project on Rational® Test Automation Server. See [Adding a performance agent to a project on page 213](#).



Note: If you want to run an AFT Suite (that contains Web UI tests) on a performance agent, you must start the performance agent as a non-admin user so that certain browsers that are configured start correctly during the test run time.

Before you add the tests to the remote repository, you must have completed the following tasks:

- Associated the performance agents with the performance schedule in Rational® Performance Tester.
- Specified the host name of the remote workstation in the AFT XML file.

You can continue to configure a run for any of the following supported test types:

- [Configuring an AFT Suite run on page 255](#)
- [Configuring a run of a Compound Test that contains performance tests or a Performance test on page 331](#)
- [Configuring a run of a Compound Test that contains Web UI tests or a Functional Web UI test on page 291](#)
- [Configuring a run of a Compound Test that contains HTML tests on page 280](#)
- [Configuring a run of a Rate Schedule or VU Schedule on page 371](#)

Viewing performance agents that are registered with Rational® Test Automation Server

After you install the remote performance agents and configure them with your offline user token, the performance agents register with Rational® Test Automation Server and you can view the registered performance agents on the **Agents and Intercepts** page.

Before you begin

You must have completed the following tasks:

- Installed the performance agent of the same version as the version of Rational® Test Automation Server.
- Configured the performance agent with your offline user token so that the performance agent can register with Rational® Test Automation Server.



Note: When you configure the performance agent to connect to the server, you must specify the offline user token that you generated on Rational® Test Automation Server. This token defines your ownership over the agent.

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Click **Infrastructure > Agents and Intercepts** in the navigation pane.

The **Agents and Intercepts** page is displayed.

You can view the Dockers, intercepts, Kubernetes clusters, or performance agents that are registered with Rational® Test Automation Server.

3. You can view the agents that you own in any of the following ways:

- Search for the agent by entering the name of the agent in the **Search** field.



Note: You can enter either the full name or any text that is in the name. The search is enabled for case sensitive text that you can enter.

- Sort the **Type** column to sort the items and display the agents in the rows at the top of the table.
- Sort the **Agents** column and identify the agents by their names or by their owner.

You can view the following details about the agents that are registered with Rational® Test Automation Server:

- The projects to which the agent is added, are displayed in the **Projects** column.
- The status of the agent is displayed in the **Status** column.



Note: You can add agents that are in the `Ready` state to your project, and then use them as locations to run tests or virtual services.

Results

You have viewed the registered agents from the **Agents and Intercepts** page.

What to do next

You can perform any of the following tasks:

- You can add the performance agents that you own to your project. See [Adding a performance agent to a project on page 213](#).
- You can add the API agents that you own to your project. See [Adding a performance agent to a project on page 213](#).

Adding a performance agent to a project

If you have performance tests that distribute workload across different workstations or Accelerated Functional Testing (AFT) Suites that need to run on different platforms and browsers, you can add IBM® Rational® Performance Tester Agents to your projects to run tests on remote workstations.

Before you begin

You must have completed the following tasks:

- Read and completed the tasks mentioned in [Test run considerations for running tests on remote performance agents on page 211](#), if you want to use a remote agent as the location to run the test.
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Installed the agent by using your offline user token that makes you the owner of the agent or you are assigned the role of a *Team Space Owner*.

About this task

Only you can add or remove a performance agent that you own from the project. However, any member of your project with the permission to run the tests can initiate the run of test assets that are associated with the performance agent.

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open your project, and then click **Infrastructure > Agents and Intercepts**.

The **Agents and Intercepts** page is displayed.

3. Click **Add > Add agent**.

The performance agents that you configured with your user token are displayed.



Note: You can add the performance agents that you own and that are in the *Ready* state. You can add any number of performance agents to your project.

4. Select the performance agents that you want to use in the project, and then click **Add**.

The performance agents that you added to the project are displayed.

5. Perform the following steps to view details and capabilities of the performance agents:
 - a. Expand the performance agent to view the details panel.

You can find the following details of the performance agent displayed:

Parameter	Description
Version	The version of the installed performance agent.
Address	The IP address of the computer on which the performance agent is installed.
License type	The license type that is enabled for you to use the performance agent.
Load generation service	The status of the performance agent whether it can be used for load generation service.
Last contact	The time of the last contact with the performance agent by Rational® Test Automation Server.

- b. Expand the capabilities panel to view the system and application capabilities.

You can find the following capabilities of the performance agent displayed:

Capabilities		Description
System capabilities	Architecture	The architecture of the CPU processor of the computer on which the performance agent is installed.
	MAC Address	The MAC address of the computer on which the performance agent is installed.

Capabilities		Description
	Operating system	The operating system of the computer on which the performance agent is installed.
	RAM	The memory that is allotted to RAM in the computer on which the performance agent is installed.
Application capabilities	Browser	The browsers along with their version that are installed on the computer on which the agent is installed.
User Defined capabilities	User Defined	The capabilities that you add for a performance agent. You can also tag a performance agent so that you can identify the performance agent to select as a location.

c. Check for the status of the performance agent from the **Status** column.



Note: You can add performance agents that are in any state to the project.

Results

You have added remote performance agents to your project and viewed the performance agent capabilities that is displayed for each agent that you own.

What to do next

You can perform any of the following tasks:

- Run the tests on the performance agent that is added to the project from the **Execution** page. See [Test run configurations on page 249](#).
- Define the capabilities that you want to add, and then view, edit, or delete the capabilities that you defined for the performance agents in your project. See [Working with the capabilities of a performance agent on page 215](#).

Working with the capabilities of a performance agent

When you installed performance agents V10.1.2 or later, and added them to your project, you can view capabilities of the performance agents from the **Infrastructure** page. When you installed performance agents V10.2.0 or later, you can add, view, edit, or delete properties and capabilities of performance agents from the **User Defined** panel of that performance agent.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Installed the agent by using your offline user token that makes you the owner of the agent or you are assigned the role of a *Team Space Owner*.
- Installed a performance agent. See [Management of performance agents on page 210](#).
- Added performance agents to your project. See [Adding a performance agent to a project on page 213](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open your project, and then click **Infrastructure > Agents and Intercepts**.

The agents that you added to your project are displayed on the **Agents and Intercepts** page.

3. Expand the performance agent.


The **Details** and **Capabilities** tabs are displayed.






4. Click **Capabilities** to view the capabilities of the performance agent.

5. Click **User Defined**.

The **User Defined Capabilities** dialog is displayed.

6. Complete the steps for the task that you want to perform as listed in the following table:

Task	Action
Adding capabilities	<p>Perform the following actions:</p> <ol style="list-style-type: none"> Click Add new. Enter the name of the capability in the Capability Name field. Perform any of the following actions in the User Defined Category field: <ul style="list-style-type: none"> ▪ Select an existing category from the list. ▪ Enter a name of the category for the capability, if the category does not exist. <p> Note: You can also click Add new to add a category.</p> <p>The category for the capability is either selected or created and the fields for the property name and value are displayed.</p>

Task	Action
	<p> Note: The property for a capability is a <i>name-value</i> pair.</p> <p>d. Enter a name for the property in the Property Name field and its corresponding value in the Property Value field for the capability.</p> <p> Note: You can add multiple properties for the same capability under the same category or under different categories by clicking Add new property, and then by entering the name and value for the property.</p> <p>e. Click Save.</p> <p>The capabilities that you added are displayed.</p>
Viewing capabilities that you added	The User Defined Capabilities panel displays the capabilities that you added to the performance agent.
Editing capabilities that you added	<p>Perform the following actions:</p> <p>a. Click the Edit icon  in the row of the capability that you want to edit.</p> <p>The user-defined capability is enabled for editing. You can edit the name of the capability, or the property name or property value.</p> <p>b. Edit any of the items that you want for the user-defined capability.</p> <p>c. Click Update.</p> <p>The updated capability of the performance agent is displayed.</p>
Deleting capabilities that you added	<p>a. Click the Delete icon  in the row of the capability that you want to delete.</p> <p>b. Click Delete.</p> <p>The capability that you selected is deleted along with all the properties that you added for the capability.</p>
Deleting a property defined for a capability	<p>a. Click the Edit icon  in the row of the capability that you want to edit and remove a defined property.</p> <p>The user-defined capability is enabled for editing.</p> <p>b. Select and delete the contents in the Property Name field and its corresponding Property Value field.</p> <p>c. Click Update.</p> <p>The property is removed for the capability.</p>

Results

You have added, viewed, edited, or deleted a property or the user-defined capability of a performance agent.

What to do next

You can run the tests on the performance agent from the **Execution** page. See [Test run configurations on page 249](#).

Management of Docker hosts

You can find information about the tasks that you can perform on remote Docker hosts. After you set up a remote Docker host, you can register the Docker host with IBM® Rational® Test Automation Server. You must then add the registered remote Docker host to your project before you can run tests on the remote Docker host location.

You can find the following information about managing Dockers:

- [Test run considerations for running tests on remote Docker hosts on page 218](#)
- [Setting up a remote Docker host computer on page 220](#)
- [Setting up a secure remote Docker host computer on page 222](#)
- [Copying third-party application Jars to a remote Docker host on page 223](#)
- [Registering a remote Docker host on page 225](#)
- [Viewing remote Docker hosts that are registered with IBM Rational Test Automation Server on page 227](#)
- [Adding a remote Docker host to the project for running tests on page 228](#)
- [Editing configurations of a remote Docker host on page 229](#)
- [Removing a remote Docker host from a project on page 232](#)
- [Unregistering a remote Docker host from IBM Rational Test Automation Server on page 233](#)

Test run considerations for running tests on remote Docker hosts

Before you can configure a test to run on a remote Docker host, you must read the considerations that you must take into account.

You can run all the tests that are supported on Rational® Test Automation Server on remote Docker hosts.



Important: You can run virtual services only in the **Default Cluster** location of Rational® Test Automation Server. You cannot run virtual services on a remote Docker host.

When you plan to run any of the tests on a remote Docker host from a project on Rational® Test Automation Server, you must complete the following tasks:

	Tasks	More information...
1.1	Set up non-secure remote Docker hosts.	Setting up a remote Docker host computer on page 220
1.2	Set up secure remote Docker hosts.	Setting up a secure remote Docker host computer on page 222

	Tasks	More information...
2	Register the remote Docker host on Rational® Test Automation Server.	Registering a remote Docker host on page 225
3	View the registered Docker hosts.	Viewing remote Docker hosts that are registered with IBM Rational Test Automation Server on page 227
4	Add a remote Docker host to the Rational® Test Automation Server project.	Adding a remote Docker host to the project for running tests on page 228
5	<p>Configure a run for your test that you want to run on the remote Docker host by performing the following actions:</p> <ol style="list-style-type: none"> 1. Go to Test run configurations on page 249. 2. Check for the prerequisite tasks that you must perform before you configure a test run for the test. 3. Select the task for the test that you want to configure for a run, and then follow the steps to run the test. 	<ol style="list-style-type: none"> 1. Test run configurations on page 249 2. Prerequisites to running tests on page 198 3. Test run configurations on page 249

If you plan to run API Suites that use a transport and the transport requires third-party application `JAR` files for a successful run on a remote Docker host, then you must perform the following task:

- You must copy the third-party application `JAR` files to the third-party application folder on the computer where the remote Docker host is installed. See [Copying third-party application Jars to a remote Docker host on page 223](#).

After you set up and register the remote Docker host with Rational® Test Automation Server, you can add the remote Docker hosts to your project on Rational® Test Automation Server. See [Adding a remote Docker host to the project for running tests on page 228](#).

You can find information about the tasks that you can perform when you use remote Docker hosts that are set up and registered with Rational® Test Automation Server.

- When you configure a test run and in the step in which you select the remote Docker, if you encounter any of the following conditions because these were changed on the remote Docker host:
 - The host name or port of the remote Docker is not displayed correctly.
 - You want to change the mode of authentication with the remote Docker host.

You must edit the configuration settings for the remote Docker host. See [Editing configurations of a remote Docker host on page 229](#).

- When you want to delete a remote Docker host from your Rational® Test Automation Server project, see [Removing a remote Docker host from a project on page 232](#).
- When you want to remove or unregister a remote Docker host from Rational® Test Automation Server, see [Unregistering a remote Docker host from IBM Rational Test Automation Server on page 233](#).

Related information

[Test run configurations on page 249](#)

[Management of Docker hosts on page 218](#)

Setting up a remote Docker host computer

You must set up a computer to host the remote Docker host. You can set up the remote host with a non-secure mode of connection with IBM® Rational® Test Automation Server.

Before you begin

You must have identified a remote computer or VM that has a minimum of 16 GB RAM. You must have installed Ubuntu V18.04 or later, on the remote computer.

You must have installed Docker Enterprise Edition for a non-secure mode of connection with IBM® Rational® Test Automation Server on the remote Docker host computer. Refer to [Get Docker Engine](#).

You must have the entitlement key that is required for you to connect to the IBM Docker images library to be able to pull in the software image to the remote Docker host.

About this task

You must set up a remote computer or a Virtual Machine (VM) on which you want to set up the remote Docker host. You must be a system administrator or get the remote Docker host set up by a system administrator. You must ensure that the system administrator completes the following actions:

- Enables SSH.
- Creates user credentials for you with root permissions.
- Provides the name of the Docker container.
- Provides the IP address of the remote Docker host computer.

1. As a system administrator of the remote system, specify a port to access the Docker daemon in the `docker.service` file located in the directory `/lib/systemd/system`. You must add the configured port at the line in the file:

```
ExecStart=/usr/bin/dockerd -H fd:// -H tcp://0.0.0.0:PORT
```

For example, if the port configured is 4342, then the line must read as:

```
ExecStart=/usr/bin/dockerd -H fd:// -H tcp://0.0.0.0:4342
```


2. Restart the Docker service by using the following commands:

```
$ sudo systemctl daemon-reload
```

```
$ sudo systemctl restart docker
```

3. Perform the following steps as a user with root permissions or ask the system administrator to complete the following steps:

a. Open a terminal and connect to the remote Docker host by using an SSH tool by running the following command:

```
ssh root@<IP_address_of_remote_host>
```

b. Log in to IBM Docker images library by running the following command:

```
sudo docker login cp.icr.io -u <username> -p <password>
```



Note: You must provide the following username and password in the command to log in to the registry:

- Username as cp
- Password as <entitlement_key>

c. Run the following commands to pull the product images from the registry on to the remote Docker host computer:



Note: You must pull in the images with the version which is the same as the version of IBM® Rational® Test Automation Server that you use to run the tests.

- `sudo docker pull cp.icr.io/cp/<ibm-rtw_docker_image>`
- `sudo docker pull cp.icr.io/cp/<ibm-rtvs_docker_image>`

Results

You have set up the remote Docker host computer that can be accessed in a non-secure mode.

What to do next

You can set up a secure remote Docker host computer optionally, that enables a secure connection with Rational® Test Automation Server. See [Setting up a secure remote Docker host computer on page 222](#).

You can register the remote Docker host on Rational® Test Automation Server. See [Registering a remote Docker host on page 225](#).

Setting up a secure remote Docker host computer

You must set up a computer to host the remote Docker host. You can set up the remote host with a secure mode of connection with IBM® Rational® Test Automation Server by using certificates issued by the remote host computer that authenticate the connection.

Before you begin

You must have identified a remote computer or VM that has a minimum of 16 GB RAM. You must have installed Ubuntu V18.04 or later, on the remote computer.

You must have installed Docker Community Edition V18.09 or later. For instructions, refer to [Get Docker Engine](#).



Note: For the remote Docker host to be reached in a safe manner through the network, you can enable *TLS* by specifying the **tlsverify** flag and by pointing the **tlscacert** flag of the remote Docker host to a trusted CA certificate. Refer to [Protect the Docker daemon socket](#).

You must have the entitlement key that is required for you to connect to the IBM Docker images library to be able to pull in the software image to the remote Docker host.

About this task

You must set up a remote computer or a Virtual Machine (VM) on which you want to set up the remote Docker host. You must be a system administrator or get the remote Docker host set up by a system administrator. You must ensure that the system administrator completes the following actions:

- Enables SSH.
- Creates user credentials for you with root permissions.
- Provides the name of the Docker container.
- Provides the IP address of the remote Docker host computer.

Perform the following steps as a user with root permissions or ask the system administrator to complete the following steps:

- a. Open a terminal and connect to the remote Docker host by using an SSH tool by running the following command:

```
ssh root@<IP_address_of_remote_host>
```

- b. Log in to IBM Docker images library by running the following command:

```
sudo docker login cp.icr.io -u <username> -p <password>
```



Note: You must provide the following username and password in the command to log in to the registry:



- Username as cp
- Password as <entitlement_key>

c. Run the following commands to pull the product images from the registry on to the remote Docker host computer:



Note: You must pull in the images with the version which is the same as the version of IBM® Rational® Test Automation Server that you use to run the tests.

- `sudo docker pull cp.icr.io/cp/<ibm-rtw_docker_image>`
- `sudo docker pull cp.icr.io/cp/<ibm-rtvs_docker_image>`

Results

You have set up the remote Docker host computer that can be accessed in a secure mode.

What to do next

You can optionally, set up a non-secure remote Docker host. See [Setting up a remote Docker host computer on page 220](#).

You can register the remote Docker host on Rational® Test Automation Server. See [Registering a remote Docker host on page 225](#).

Copying third-party application Jars to a remote Docker host

You can run API Suites in a project on Rational® Test Automation Server on a remote Docker host. If the API Suites use a transport and the transport requires third-party `Jar` files for a successful run, you must ensure that the third-party application `Jar` files are available at the test run time. To achieve this, you must copy the third-party application `Jar` files to the computer where you have set up the remote Docker host.

Before you begin

If you want to copy the third-party application `Jar` files to the to the computer where Rational® Test Automation Server is installed on Kubernetes, see [Copying third-party application Jars to Kubernetes on page 118](#).

You must have server administrator privileges.

You must have completed the following tasks:

- Set up and configured the remote Docker host. See [Management of Docker hosts on page 218](#).
- Identified the third-party application `Jar` files that are required and copied the `Jar` files. See [Test run considerations for API Suites or API tests on page 200](#).
- Copied the `Jar` files of the third-party application jars to the directory or folder on the remote Docker host from where you can run the docker commands.

About this task

You can copy the required third-party application `jar` files to the folder that is specific for the application under the `/myFiles/Userlibs/<application_name>` folder. You need not extract the files.

You can perform this task any time after you have configured the remote Docker host and you plan to run an API Suite on the remote Docker host. The API Suite uses transports and the transports require third-party application `JAR` files to be available at the test run time.

You can get help on the docker commands by running the command: `$ docker help` from the docker command line. For more information about the docker commands, refer to the [Docker command line documentation](#).

1. Use the following table to find the name of the folder that corresponds to the specific third-party application for the transport used in the API Suite.



Note: You must provide the name of the folder listed for the third-party application as the `<application_name>` in the docker command.

Table 5. Name of the folder for the application

Application	Name of the folder to use
Camel	Camel
CentraSite	CentraSite
CICS	CICS
Coherence	Coherence
Database	JDBC
IMS	IMS
Integra	Integra
JMS	JMS
SAP RFC	SAP
Software AG Universal Messaging	SoftwareAGUM
TIBCO EMS	TIBCO
TIBCO Rendezvous	
TIBCO SmartSockets	
WebSphere Application Server Service Integration Bus (SiBus)	WAS
WebLogic	WebLogicJMX

Application	Name of the folder to use
Software AG webMethods	webMethods

- Run the following docker command to create or update the volume that contains the application `jar` files:

```
docker run --rm -v /<anyFolder>/UserLibs/<application_name>:/ulsrc -v userlibs/<application_name>:/uldest
alpine:latest cp -r /ulsrc/. /uldest/UserLibs/<application_name>
```



Attention: You must run the docker command when no other tests are running in the Docker host to prevent concurrent access problems.

Results

You have successfully copied the third-party application `jar` files to the folder in the remote Docker host.

What to do next

- You can register the remote Docker host, if you have not already done so. See [Registering a remote Docker host on page 225](#).
- You can configure the API Suite run. See [Configuring an API Suite or an API test run on page 267](#).

Registering a remote Docker host

You must register the remote Docker host with Rational® Test Automation Server before you add the remote Docker host to your project in a team space, which contains the test assets that you want to run on the remote Docker host.

Before you begin

Depending on the mode of connection with Rational® Test Automation Server, you must have set up the remote Docker host computer either for a secured mode or for an unsecured mode of connection.

- Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

- Click **Infrastructure > Agents and Intercepts** in the navigation pane.

The **Agents and Intercepts** page is displayed.

You can view the agents, intercepts or Dockers that are registered with Rational® Test Automation Server.

- Click **Add new > Add new docker host**.

The **New docker host** dialog is displayed.

- Enter the *host name* and the *port* of the remote Docker host in the format `<host name or IP_address>:<port>` in the **Remote Docker Host** field.

If you do not provide the port that you configured for the remote Docker host, then the default port that is used is as follows:

- Port 2375 for the unsecured mode
- Port 2376 for the secured mode

5. Choose from any of the following authentication modes to establish the connection between Rational® Test Automation Server and the remote Docker host computer:

- For the secured mode, which is the default mode and the checkbox is displayed as selected.

Perform the following steps:

- a. Select **Secure mode**, if it is not selected.
- b. Click the action labels to browse and select the `.pem` files that correspond to `ca_certificate.pem`, `client_certificate.pem`, and `client_key.pem`.



Note: You must have generated the `.pem` files on the remote Docker host computer and copied the files to your local drive.

- For the unsecured mode, the **Secure mode** checkbox must not be selected.

Perform the following step:

- a. Clear the **Secure mode** checkbox, if selected.

6. Click **Test connection** to test if a connection is established between Rational® Test Automation Server and the remote host computer.

Any of the following events occur when **Test connection** is clicked:

- On a successful connection, a message is displayed.
- On a failure to connect to the remote host computer, an error message is displayed. You must resolve the error and reattempt to establish a successful connection.



Important: The remote Docker host computer must be connected successfully before you register the remote Docker host with Rational® Test Automation Server.

7. Click **Register**.

The remote Docker host that you registered is displayed.



Important: The remote Docker host must be registered successfully with Rational® Test Automation Server in the team space before you add the remote Docker host to your project in that team space.

8. Click **Close** to exit.

Results

You have registered the remote Docker host in your team space with Rational® Test Automation Server.

What to do next

You can view the Docker hosts in your team space that are registered with Rational® Test Automation Server. See [Viewing remote Docker hosts that are registered with IBM Rational Test Automation Server on page 227](#).

You must add the registered remote Docker host to your project in your team space before you use the remote Docker host as a location to run tests in your project. See [Adding a remote Docker host to the project for running tests on page 228](#).

Viewing remote Docker hosts that are registered with IBM® Rational® Test Automation Server

You must register the remote Docker host in your team space with Rational® Test Automation Server before you add the remote Docker host to your project in that team space, which contains the test assets that you want to run on the remote Docker host.

Before you begin

Depending on the mode of connection with Rational® Test Automation Server, you must have set up the remote Docker host computer either for a secured mode or for an unsecured mode of connection. See [Setting up a remote Docker host computer on page 220](#) or [Setting up a secure remote Docker host computer on page 222](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Click **Infrastructure > Agents and Intercepts** in the navigation pane.

The **Agents and Intercepts** page is displayed.

You can view the agents, intercepts or Dockers that are registered with Rational® Test Automation Server in your team space.

3. You can view the Dockers that you own in any of the following ways:

- Search for the Docker by entering the name of the Docker host in the **Search** field.



Note: You can enter either the full name or any text that is in the name. The search is enabled for case sensitive text that you can enter.

- Sort the **Type** column to sort the items and then identify the Docker by the name displayed.
- Sort the **Agents** column to sort the items and identify the Docker by the owner.

You can view the following details about the Dockers that are registered with Rational® Test Automation Server in the team space:

- The projects in the team space to which the Docker host is added, are displayed in the **Projects** column.
- The status of the Docker host is displayed in the **Status** column.



Note: You can add Docker hosts that are in the `Ready` state to your project, and then use them as locations to run tests.

Results

You have viewed the remote Docker hosts that are registered with Rational® Test Automation Server in your team space.

What to do next

You must add the registered remote Docker host to your project in your team space before you use the remote Docker host as a location to run tests in your project. See [Adding a remote Docker host to the project for running tests on page 228](#).

Adding a remote Docker host to the project for running tests

You can choose to run tests on Docker hosts that you have set up on remote host computers. You must register the Docker host with IBM® Rational® Test Automation Server and then add them to your project before you run tests on the remote Docker hosts.

Before you begin

You must have completed the following tasks:

- Read about the considerations you must take into account before you configure a test run to run on a remote Docker host. See [Test run considerations for running tests on remote Docker hosts on page 218](#).
- Set up the remote Docker host system. See [Setting up a remote Docker host computer on page 220](#) or [Setting up a secure remote Docker host computer on page 222](#).
- Registered the remote Docker host with IBM® Rational® Test Automation Server. See [Registering a remote Docker host on page 225](#).

1. Log in to Rational® Test Automation Server.
2. Open your project.
The **Overview** page is displayed.
3. Click **Infrastructure > Agents and Intercepts**.

The **Agents and Intercepts** page is displayed.

4. Click **Add > Add docker**.
5. Select the Docker host that you want to add to the project from the list of Docker hosts, and then click **Add**.



Note: You can add the Docker hosts that you registered. You can add any number of Docker hosts to your project.

The Docker hosts that you added to the project are displayed.

You can view the details of the Docker host by clicking the **Expand** icon

The screenshot shows the 'Agents' page for a project named 'proj1'. At the top, there is a search bar with the text 'Search'. Below it, there is a table with the following columns: 'Type' and 'Agents'. The table contains one entry with the IP address '10.121.104.38:80' and the text 'Owned by me'. To the left of this entry is a dropdown menu with a checkmark and a trash icon. Below the table, there is a 'Details' section for the selected agent, showing the following information:

Server:	Docker Engine - Community
Secure	No
Version:	19.03.2
Address:	10.121.104.38
Port:	80
Operating system:	linux
Architecture:	amd64
Api version:	1.40 (minimum version 1.12)
Go version:	go1.12.8
Git commit:	6a30dfc
Built:	Thursday, August 29, 2019 5:27 AM
Experimental:	false

What to do next

You can select any of the remote Docker hosts as an alternate location to run the test asset in your project while configuring a test run from the **Execution** page.

Related information

[Test run configurations on page 249](#)

Editing configurations of a remote Docker host

You can edit the host name or the port of the registered Docker host computer instead of registering it as a new remote host if any of the parameters are changed on the remote host computer. You can also change the mode of authentication of a registered remote Docker host.

Before you begin

You must have completed the following tasks:

- Set up the remote Docker host computer. See [Setting up a remote Docker host computer on page 220](#) or [Setting up a secure remote Docker host computer on page 222](#).
- Registered the remote Docker host with IBM® Rational® Test Automation Server in your team space. See [Registering a remote Docker host on page 225](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Click **Infrastructure > Agents and Intercepts** in the navigation pane.

The **Agents and Intercepts** page is displayed.

You can view the agents, intercepts or Dockers that are registered with Rational® Test Automation Server in your team space.

3. You can view the Dockers that you own in any of the following ways:

- Search for the Docker by entering the name of the Docker host in the **Search** field.




Note: You can enter either the full name or any text that is in the name. The search is enabled for case sensitive text that you can enter.


- Sort the **Type** column to sort the items and then identify the Docker by the name displayed.
- Sort the **Agents** column to sort the items and identify the Docker by the owner.

You can view the following details about the Dockers that are registered with Rational® Test Automation Server in your team space:

- The projects in the team space to which the Docker host is added, are displayed in the **Projects** column.
- The status of the Docker host is displayed in the **Status** column.

4. Identify the Docker host that you want to modify the configuration, and then click the **Edit** icon .




Note: The  icon is displayed in the **Actions** column in the row of the Docker only if you own the remote Docker host.

The **Update docker host** dialog is displayed.

5. Change any of the configurations.

To change...	Do this...
Change the host name or the port of the remote Docker host.	Enter the changed value of the host name or the port in the Remote hostname field.

To change...	Do this...
Change the mode of authentication from a non-secure mode to a secure mode.	a. Select the Secure mode option. b. Click the action labels to browse and select the <code>.pem</code> files that correspond to <code>ca_certificate.pem</code> , <code>client_certificate.pem</code> , and <code>client_key.pem</code> .  Note: You must have generated the <code>.pem</code> files on the remote host computer and copied the files to your local drive.
Change the mode of authentication from a secure mode to a non-secure mode.	Clear the Secure mode option that is selected.

6. Click **Test connection** to test whether a connection is established between Rational® Test Automation Server and the remote host computer:

- On a successful connection, a message is displayed.
- On a failure to connect to the remote host computer, an error message is displayed. You must resolve the error and reattempt to establish a successful connection.

7. Click **Update**.

The remote Docker host with the updated details is displayed.

Results

You have updated the registered remote Docker host with the changed parameters.

What to do next

You can now add the updated Docker host to your project. See [Adding a remote Docker host to the project for running tests on page 228](#).

Related information

[Setting up a remote Docker host computer on page 220](#)

[Registering a remote Docker host on page 225](#)

[Removing a remote Docker host from a project on page 232](#)

[Unregistering a remote Docker host from IBM Rational Test Automation Server on page 233](#)

Removing a remote Docker host from a project

You can delete a remote Docker host that you added to your project in IBM® Rational® Test Automation Server when you no longer need it to run tests.

Before you begin

You must have completed the following tasks:

1. Set up the remote Docker host computer. See [Setting up a remote Docker host computer on page 220](#) or [Setting up a secure remote Docker host computer on page 222](#).
2. Registered the remote Docker host with Rational® Test Automation Server in your team space. See [Registering a remote Docker host on page 225](#).
3. Added the registered Docker host to a project in your team space. See [Adding a remote Docker host to the project for running tests on page 228](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project in your team space from the **My Projects** page.

The **Overview** page is displayed.

3. Click **Infrastructure > Agents and Intercepts** in the navigation pane.

The **Agents and Intercepts** page is displayed.

You can view the agents, intercepts or Dockers that are added to your project.

4. You can view the Dockers that you own in any of the following ways:

- Search for the Docker by entering the name of the Docker host in the **Search** field.




Note: You can enter either the full name or any text that is in the name. The search is enabled for case sensitive text that you can enter.

- Sort the **Type** column to sort the items and then identify the Docker by the name displayed.
- Sort the **Agents** column to sort the items and identify the Docker by the owner.

You can view the following details about the Dockers that are registered with Rational® Test Automation Server:

- The projects to which the Docker host is added, are displayed in the **Projects** column.
- The status of the Docker host is displayed in the **Status** column.

- Identify the Docker host that you want to remove from the project, and then click the **Remove docker host** icon .



Note: The  icon is displayed in the **Actions** column in the row of the Docker only if you own the remote Docker host.

A message is displayed that the Docker host is removed from the project successfully.

Results

You have removed the remote Docker host from your project. The remote Docker host is no longer available as a location while configuring a test run in your project.

What to do next

You might have to add a remote Docker host to your project to use the remote Docker host as a location to run tests.

Related information

[Unregistering a remote Docker host from IBM Rational Test Automation Server on page 233](#)

[Editing configurations of a remote Docker host on page 229](#)

Unregistering a remote Docker host from IBM® Rational® Test Automation Server

You can unregister a remote Docker host that is registered with Rational® Test Automation Server in a team space when you no longer require it.

Before you begin

You must have completed the following tasks:

- Set up the remote Docker host computer. See [Setting up a remote Docker host computer on page 220](#) or [Setting up a secure remote Docker host computer on page 222](#).
- Registered the remote Docker host with Rational® Test Automation Server in a team space. See [Registering a remote Docker host on page 225](#).

- Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.



- Click **Infrastructure > Agents and Intercepts** in the navigation pane.


The **Agents and Intercepts** page is displayed.

You can view the agents, intercepts or Dockers that are registered with Rational® Test Automation Server in your team space.

3. Identify the Docker host that you want to unregister.



Note: The  and  icons are displayed in the **Actions** column in the row of the Docker only if you own the remote Docker host.

4. Unregister the Docker host by completing the following steps:
 - a. Click the **Unregister docker host** icon .
 - b. Click **Unregister** in the **Unregister docker host** dialog box.

A message is displayed that the Docker host is unregistered successfully.

Results

You have unregistered the remote Docker host in a team space on Rational® Test Automation Server. You cannot run tests on this remote Docker host if it is added to your project. You might have to register it again with Rational® Test Automation Server, if you want to use the remote Docker host in your projects.

Related information

[Editing configurations of a remote Docker host on page 229](#)

[Adding a remote Docker host to the project for running tests on page 228](#)

[Removing a remote Docker host from a project on page 232](#)

Management of remote Kubernetes clusters

You can find information about the tasks that you can perform on remote Kubernetes clusters. After you set up a remote Kubernetes cluster, you can register the external cluster with a team space on IBM® Rational® Test Automation Server. You must then add the remote cluster to a project before you run test assets or resources on the remote Kubernetes cluster.

You can find the following information about managing the external Kubernetes cluster:

- [Considerations for using a remote Kubernetes cluster on page 235](#)
 - [Registering a remote Kubernetes cluster on page 237](#)
 - [Viewing remote Kubernetes clusters on page 240](#)
 - [Adding a remote Kubernetes cluster to a project on page 242](#)
 - [Editing configurations of a remote Kubernetes cluster on page 244](#)
 - [Removing a remote Kubernetes cluster from a project on page 246](#)
 - [Unregistering a remote Kubernetes cluster from a team space on page 248](#)
-

Considerations for using a remote Kubernetes cluster

When you want to use a remote Kubernetes cluster as a location to run the supported tests or test resources, you must read the considerations first.

Before you want to run any of the supported tests or test resources on Rational® Test Automation Server that are configured to run on a remote Kubernetes cluster, you must have completed the following tasks:

- Verified that the Kubernetes cluster that you want to use is a supported one by referring to the OS, platform, and containers that are specified for Rational® Test Automation Server in [System Requirements on page 12](#).
- Kept a suitable `kubeconfig` file with access to the remote Kubernetes cluster in a folder on the computer that you want to use to access an instance of Rational® Test Automation Server.

You must be familiar with the administration of a Kubernetes cluster. When you as the cluster administrator, set up the remote Kubernetes cluster must consider the following requirements:

- You must have a `kubeconfig` file with access to the remote Kubernetes cluster in a folder on the computer that you want to use to access an instance of Rational® Test Automation Server.



Note: If the `kubeconfig` file references certificate files by using filepaths then the certificate data must be embedded in the `kubeconfig` file to make it usable by Rational® Test Automation Server. If the `kubeconfig` file contains multiple contexts, then only the current context is required.

You can use the following `kubectl` command to create a suitable `kubeconfig` file called `portable-config` in the current folder. The `portable-config` file contains embedded certificate data and the information used by the current context:

```
kubectl config view --flatten --minify > portable-config
```

- You must ensure that the remote Kubernetes cluster is able to access the Rational® Test Automation Server image registry which is the same registry that you used to install Rational® Test Automation Server. There is no need to copy the Rational® Test Automation Server image to the remote Kubernetes cluster but the remote cluster must be able to access the image registry at test runtime.

If the remote cluster that you want to use is a Lightweight Kubernetes (k3s) cluster, it might require a `Secret` to pull the Rational® Test Automation Server image. You must run the following command on the remote k3s cluster:

```
kubectl create secret docker-registry cp.icr.io \
-n {remote-namespace} \
--docker-server=cp.icr.io \
--docker-username=cp \
--docker-password={your_entitlement_key} \
--docker-email={example@abc.com}
```



Notes:



- You must replace *{remote-namespace}* with the value of the *namespace* that you have used for the remote cluster.
 - If the username contains any special characters, such as \$, you must enclose it within single quotation-marks.
 - You can replace *{example@abc.com}* with the email address of the administrator, if required.
- You need not configure separate licenses for running on the remote cluster because the licenses are consumed from the pool of available licenses that are configured for the team space on Rational® Test Automation Server. The licenses are returned to the pool when the test run is completed on the remote cluster.
 - You must ensure that the Rational® Test Automation Server domain is accessible from the remote cluster so that the pod in the cluster is able to communicate with Rational® Test Automation Server.
 - You must ensure that the appropriate **PriorityClass** exists on the remote cluster if you want to control the priority given to the remote execution pod by the scheduler. The same **PriorityClass** that is used by the Rational® Test Automation Server execution pod is used by the remote execution pod. If the named **PriorityClass** does not exist in the remote cluster, then no **PriorityClass** is specified by the remote execution pod.
 - You must provide the Ingress details such as Ingress type, Ingress host pattern, and Ingress secret when you register a remote cluster with your team space on Rational® Test Automation Server. These configurations are required when you want to run virtual services on the remote cluster and run tests from the remote cluster that use static agents.
 - You can enter the code snippets for *nodeSelectors* and *tolerations* in a `yaml` format, if you want to use *nodeSelectors* and *tolerations* on the remote cluster. You can enter these in the fields provided in the **Advanced** configuration dialog when you register a remote cluster with your team space on Rational® Test Automation Server.

For more information about *nodeSelector*, see [nodeSelector](#) and for *tolerations*, see [Tolerations](#).



Restrictions: The following restrictions apply when you want to run the supported tests on a remote Kubernetes cluster:

- Using the **UserLibs** that are required for running certain API Suites are not supported on the remote cluster.
- Running of Compound Tests that contain performance tests, which require the **NodePort** access to connect to static agents, are not supported on the remote cluster.

When you plan to run any of the test assets or resources on a remote Kubernetes cluster from a project on Rational® Test Automation Server, you must complete the following tasks:

	Tasks	More information...
1	Register a remote Kubernetes cluster with a team space on Rational® Test Automation Server.	Registering a remote Kubernetes cluster on page 237
2	View remote Kubernetes clusters added to a team space.	Viewing remote Kubernetes clusters on page 240
3	Add a remote Kubernetes cluster to a project in a team space.	Adding a remote Kubernetes cluster to a project on page 242
4.1	<p>Configure a run for your test that you want to run on the remote Kubernetes cluster by performing the following actions:</p> <ol style="list-style-type: none"> 1. Check for the prerequisite tasks that you must perform before you configure a test run for the test. 2. Select the task for the test that you want to configure for a run, and then follow the steps to run the test. 	<ol style="list-style-type: none"> 1. Test run configurations on page 249 2. Prerequisites to running tests on page 198 3. Test run configurations on page 249
4.2	<p>Configure a run of the virtual service resources that you want to run on the remote Kubernetes cluster by performing the following actions:</p> <ol style="list-style-type: none"> 1. Check for the prerequisite tasks that you must perform before you configure a run of the virtual services. 2. Select the virtual service resources in your project that you want to configure for a run, and then follow the steps to run them. 	<ol style="list-style-type: none"> 1. Prerequisites for running virtual services on page 410 2. Configuring a run of a virtual service on page 419

Related information

[Test run configurations on page 249](#)

[Management of remote Kubernetes clusters on page 234](#)

Registering a remote Kubernetes cluster

You must register a remote Kubernetes cluster to your team space on Rational® Test Automation Server before you add the remote cluster to your project.

Before you begin

You must have completed the following tasks:

- Verified that the Kubernetes cluster that you want to use is a supported one by referring to the OS, platform, and containers that are specified for Rational® Test Automation Server in [System Requirements on page 12](#).
- Kept a suitable `kubeconfig` file with access to the remote Kubernetes cluster in a folder on the computer that you want to use to access an instance of Rational® Test Automation Server.
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Click **Infrastructure > Agents and Intercepts** in the navigation pane of the team space.

The **Agents and Intercepts** page is displayed.

3. Click **Add new > Add new cluster**.

The **New cluster** dialog is displayed.

4. Perform the following steps to add the `kubeconfig` file:

a. Click **Browse**, and locate the `kubeconfig` file of the Kubernetes cluster that you want to add.

b. Select the `kubeconfig` file, and then click **Open**.

The name of the `kubeconfig` file that you selected is displayed along with the address of the Kubernetes cluster that is specified in the file.

c. Click **Next**.

The address of the Kubernetes cluster that is configured in the `kubeconfig` file is displayed in the **Address** field.



Note: You can change the address of the Kubernetes cluster if it is different from the address in the `kubeconfig` file.

5. Perform the following steps to add a namespace and name for the cluster:

a. Enter the namespace of the cluster in the **Namespace** field.

b. Enter a name for the cluster in the **Name** field.

The name helps you to identify the cluster.

c. Click **Next**.

6. Perform the following steps to add the ingress details of the cluster:

a. Select the ingress type of the cluster from the **Ingress type** list.

You can select the ingress type from the following options that are displayed:

Option	Description	Result
None	When selected specifies that no ingress type is configured for the remote Kubernetes cluster.	You can run any of the supported tests on the remote cluster but not on the remote agents.
NGINX	When selected specifies that NGINX is configured as the ingress type for the remote Kubernetes cluster.	You can run any of the supported tests on the remote cluster or on any of the remote agents.

b. Enter the ingress host pattern of the cluster in the **Ingress host pattern** field.

The DNS used by a computer that runs a client application (such as a static agent) or the HTTP proxy that routes traffic to the virtual service needs to resolve such host names to the IP address of the ingress domain.

For example, if the IP of the remote cluster is 12.12.12.123, then you must enter the ingress host pattern as `*.12.12.12.123.nip.io`

The *Unique ID* of the virtual service is prefixed to the ingress host name when virtual services are routed from the Rational® Test Automation Server cluster.

c. Enter the name of the secret in the remote namespace, which contains the certificate information.

You must have created the secret and named it following the naming conventions per the Kubernetes guidelines. The certificate that you create is a wildcard certificate which contains the Ingress host pattern of the Kubernetes cluster. This wildcard certificate ensures that the client applications can trust the virtual services.



Note: If a secret name is not provided then the default certificate that is provided by NGINX is used, which might cause issues to authenticate the virtual service by the client applications.

d. Click **Next**.

7. Optionally, perform the following steps to enter the code snippets for *nodeSelectors* and *tolerations*, if applicable for the remote cluster:

a. Enter the code snippet for *nodeSelectors* in a `yaml` format in the **Node selectors** field.


b. Enter the code snippet for *tolerations* in a `yaml` format in the **Tolerations** field.

c. Select the **Copy pull secret to the cluster** checkbox, if it is not already selected.



Note: The remote Kubernetes cluster pulls the secret to access the Rational® Test Automation Server image registry which is the same registry that you used to install Rational® Test Automation Server. The remote Kubernetes cluster must be able to access the image registry at test runtime.

d. Click **Register**.

The remote cluster is added to the team space and is displayed with the  icon in the **Type** column on the **Agents and Intercepts** page. You can identify the cluster with its name in the **Agents** column.



Note: The **Projects** column does not display any projects that are associated with the cluster yet.

Results

You have registered a remote Kubernetes cluster with your team space on Rational® Test Automation Server.

What to do next

You must add the remote cluster to your project in your team space before you use the remote cluster as a location to run test assets or resources in your project. See [Adding a remote Kubernetes cluster to a project on page 242](#).

Related information

[Management of remote Kubernetes clusters on page 234](#)

Viewing remote Kubernetes clusters

You can view the clusters that you have registered with a team space on IBM® Rational® Test Automation Server to view their details or edit their configurations.

Before you begin

You must have completed the following tasks:

- Registered a remote Kubernetes cluster with your team space on IBM® Rational® Test Automation Server. See [Registering a remote Kubernetes cluster on page 237](#).
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

- Click **Infrastructure > Agents and Intercepts** in the navigation pane of the team space.

The **Agents and Intercepts** page is displayed.

You can view the agents, intercepts, Dockers, or clusters that are registered with the team space on Rational® Test Automation Server.

- You can view the clusters that you own in any of the following ways:
 - Search for the cluster by entering the name of the cluster in the **Search** field.



Note: You can enter either the full name or any text that is in the name. The search is enabled for case-sensitive text that you can enter.


- Sort the **Type** column to sort the items, and then identify the cluster by the name or owner.
- Sort the **Agents** column to sort the items, and then identify the cluster by the name or owner.

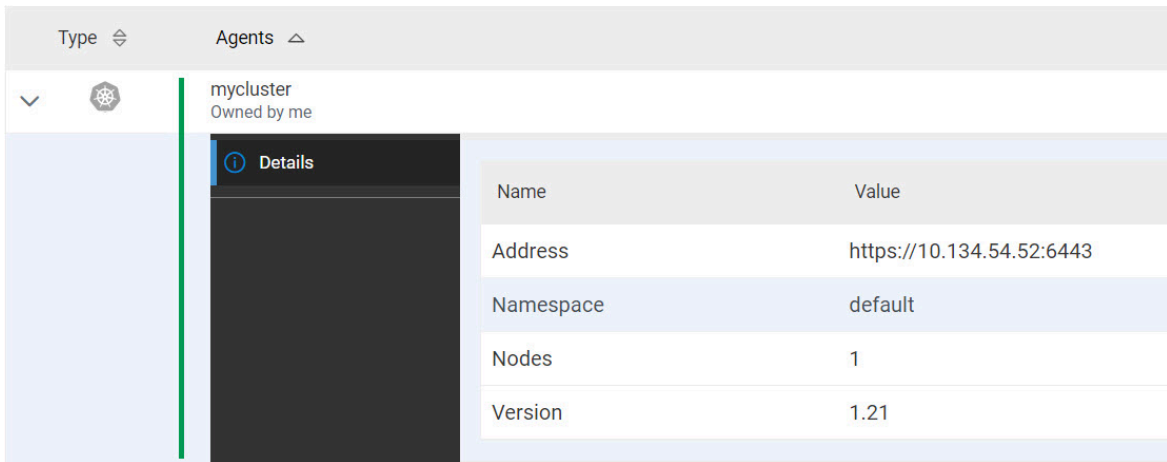
You can view the following details about the clusters that are in the team space:

- The projects in the team space to which the cluster is added are displayed in the **Projects** column.
- The status of the cluster is displayed in the **Status** column.



Note: You can add clusters that are in the `Ready` state to your project, and then use them as locations to run tests.

You can view the details of the cluster by clicking the **Expand** icon .



The screenshot shows a table with columns for 'Type' and 'Agents'. The 'Agents' column is expanded to show details for a cluster named 'mycluster', which is 'Owned by me'. A 'Details' panel is open, displaying the following information:

Name	Value
Address	https://10.134.54.52:6443
Namespace	default
Nodes	1
Version	1.21

The following table describes the details about the cluster:

Name	Description
Address	Specifies the address of the remote Kubernetes API.
Namespace	Specifies the <i>namespace</i> of the remote Kubernetes cluster in which the test runs occur.

Name	Description
Nodes	Specifies the number of nodes in the remote Kubernetes cluster.
Version	Specifies the Kubernetes version of the remote Kubernetes cluster.

Results

You have viewed details of the remote clusters that are registered with a team space on Rational® Test Automation Server.

What to do next

You must add the remote cluster to your project in your team space before you can use the remote cluster as a location to run test assets or resources that are in your project. See [Adding a remote Kubernetes cluster to a project on page 242](#).

Related information

[Management of remote Kubernetes clusters on page 234](#)

Adding a remote Kubernetes cluster to a project

You must add a remote Kubernetes cluster to your project after you register the cluster with your team space on IBM® Rational® Test Automation Server.

Before you begin

You must have completed the following tasks:

- Read and completed the tasks mentioned in [Considerations for using a remote Kubernetes cluster on page 235](#), if you want to use a remote Kubernetes cluster as a location to run the test.
- Registered a remote Kubernetes cluster with your team space on IBM® Rational® Test Automation Server. See [Registering a remote Kubernetes cluster on page 237](#).
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Been assigned a role as an *Owner* in the project.
- Created a project.

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open your project.

The **Overview** page is displayed.

3. Click **Infrastructure > Agents and Intercepts** in the navigation pane of the team space.

The **Agents and Intercepts** page is displayed.

4. Click **Add > Add cluster**.

The **Add cluster** dialog is displayed.

5. Select the cluster that you want to add to the project from the list of clusters that are registered with the team space, and then click **Add**.



Note: You can add the clusters that are registered with the team space. You can add any number of clusters to your project.

The clusters that you added to the project are displayed.

You can view the details of the cluster by clicking the **Expand** icon >.

Name	Value
Address	https://10.134.54.52:6443
Namespace	default
Nodes	1
Version	1.21

The following table describes the details about the cluster:

Name	Description
Address	Specifies the address of the remote Kubernetes API.
Namespace	Specifies the <i>namespace</i> of the remote Kubernetes cluster in which the test runs occur.
Nodes	Specifies the number of nodes in the remote Kubernetes cluster.
Version	Specifies the Kubernetes version of the remote Kubernetes cluster.

What to do next

You can select any of the remote clusters as an alternate location to run the test assets or resources that are in your project while you configure a run from the **Execution** page.

Related information

[Test run configurations on page 249](#)

[Management of remote Kubernetes clusters on page 234](#)

Editing configurations of a remote Kubernetes cluster

You can edit the `kubeconfig` file, address, namespace, name, or ingress details of the remote Kubernetes cluster that you added to a team space on IBM® Rational® Test Automation Server.

Before you begin

You must have completed the following tasks:

- Read and completed the tasks mentioned in [Considerations for using a remote Kubernetes cluster on page 235](#), if you want to use a remote Kubernetes cluster as a location to run the test.
- Registered a remote Kubernetes cluster with your team space on IBM® Rational® Test Automation Server. See [Registering a remote Kubernetes cluster on page 237](#).
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Click **Infrastructure > Agents and Intercepts** in the navigation pane of the team space.

The **Agents and Intercepts** page is displayed.

You can view the agents, intercepts, Dockers, or clusters that are registered with the team space on Rational® Test Automation Server.

3. You can view the clusters that you own in any of the following ways:

- Search for the cluster by entering the name of the cluster in the **Search** field.



Note: You can enter either the full name or any text that is in the name. The search is enabled for case-sensitive text that you can enter.


- Sort the **Type** column to sort the items, and then identify the cluster by the name or owner.
- Sort the **Agents** column to sort the items, and then identify the cluster by the name or owner.

You can view the following details about the clusters that are in the team space:


- The projects in the team space to which the cluster is added, are displayed in the **Projects** column.
- The status of the cluster is displayed in the **Status** column.



Note: You can add clusters that are in the `Ready` state to your project, and then use them as locations to run tests.

4. Identify the cluster that you want to modify the configurations, and then click the **Edit** icon .



Note: The  icon is displayed in the **Actions** column in the row of the cluster only if you own the cluster.

The **Update cluster** dialog is displayed.

5. Perform the following steps to change the `kubeconfig` file:
 - a. Click **Browse**, and locate the `kubeconfig` file of the Kubernetes cluster that you want to use.
 - b. Select the `kubeconfig` file, and then click **Open**.

The name of the `kubeconfig` file that you selected is displayed along with the address of the Kubernetes cluster that is specified in the file.

- c. Click **Next**.
6. Perform the following steps to change the namespace or name of the cluster:
 - a. Edit or change the namespace of the cluster that you want to use, in the **Namespace** field.
 - b. Edit or change the name of the cluster in the **Name** field.
 - c. Click **Next**.

7. Perform the following steps to change the ingress details of the cluster:

- a. Select the ingress type of the cluster from the **Ingress type** list, if you want to use a different cluster.
- b. Enter or change the ingress host pattern of the cluster in the **Ingress host pattern** field, if you did not provide it when setting up the remote cluster or change it to match the cluster you want to use.
- c. Enter the secret if you did not provide it when setting up the remote cluster.
The secret is used to authenticate the remote cluster to use the licenses from the Team space License Server so that the inbound traffic is allowed into the ingress domain.



Note: The secret to use must be available with your Rational® Test Automation Server server administrator.

- d. Click **Next**.

8. Optionally, perform the following steps to edit or change the code snippets for *nodeSelectors* and *tolerations*, if applicable for the remote cluster:
 - a. Enter the code snippet for *nodeSelectors* in a `yaml` format in the **Node selectors** field.
 - b. Enter the code snippet for *tolerations* in a `yaml` format in the **Tolerations** field.
9. Click **Update**.

The remote cluster is displayed with the updated configurations.



Note: The projects that are associated with the cluster are displayed in the **Projects** column and you need not add the cluster to the same project again.

Results

You have updated an existing remote cluster with the changed configurations.

What to do next

You can select the updated cluster as an alternate location to run the test assets or resources that are in your project while you configure a run from the **Execution** page.

Related information

[Management of remote Kubernetes clusters on page 234](#)

Removing a remote Kubernetes cluster from a project

You can remove a remote Kubernetes cluster from a project in a team space on IBM® Rational® Test Automation Server when you no longer need the cluster to run tests.

Before you begin

You must have completed the following tasks:

1. Registered a remote Kubernetes cluster with your team space on IBM® Rational® Test Automation Server. See [Registering a remote Kubernetes cluster on page 237](#).
2. Added the Kubernetes cluster to a project in your team space. See [Adding a remote Kubernetes cluster to a project on page 242](#).



Note: You can remove remote Kubernetes clusters that only you added to the project.

3. Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
4. Been assigned a role as an *Owner* in the project.

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project in your team space.

The **Overview** page is displayed.

3. Click **Infrastructure > Agents and Intercepts** in the navigation pane of your team space.

The **Agents and Intercepts** page is displayed.

You can view the agents, intercepts, Dockers, or clusters that are added to your project.


4. You can view the clusters that you own in any of the following ways:

- Search for the cluster by entering the name of the cluster in the **Search** field.



Note: You can enter either the full name or any text that is in the name. The search is enabled for case-sensitive text that you can enter.

- Sort the **Type** column to sort the items, and then identify the cluster by the name or owner.
- Sort the **Agents** column to sort the items, and then identify the cluster by the name or owner.

5. Identify the cluster that you want to remove from the project, and then click the **Remove cluster** icon .



Note: The  icon is displayed in the **Actions** column in the row of the cluster only if you own the remote cluster.

A message is displayed that the remote cluster is removed from the project successfully.

Results

You have removed the remote cluster from your project. The remote cluster is no longer available as a location when you configure a run of the test assets or resources that are in your project.

What to do next

You might have to add a remote cluster to your project to use the remote cluster as a location to run test assets or resources.

Related information

[Management of remote Kubernetes clusters on page 234](#)

[Unregistering a remote Kubernetes cluster from a team space on page 248](#)

[Editing configurations of a remote Kubernetes cluster on page 244](#)

Unregistering a remote Kubernetes cluster from a team space

You can unregister a remote Kubernetes cluster from a team space when you no longer require it.

Before you begin

You must have completed the following tasks:

1. Registered a remote Kubernetes cluster with your team space on IBM® Rational® Test Automation Server. See [Registering a remote Kubernetes cluster on page 237](#).
2. Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
3. Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Click **Infrastructure > Agents and Intercepts** in the navigation pane of the team space.

The **Agents and Intercepts** page is displayed.

You can view the agents, intercepts, Dockers, or clusters that are registered with the team space on Rational® Test Automation Server.

3. You can view the clusters that you own in any of the following ways:

- Search for the cluster by entering the name of the cluster in the **Search** field.





Note: You can enter either the full name or any text that is in the name. The search is enabled for case-sensitive text that you can enter.


- Sort the **Type** column to sort the items, and then identify the cluster by the name or owner.
- Sort the **Agents** column to sort the items, and then identify the cluster by the name or owner.

4. Identify the cluster that you want to remove.



Note: The  and  icons are displayed in the **Actions** column in the row of the cluster only if you own the remote cluster.

5. Unregister the cluster by completing the following steps:

- a. Click the **Unregister cluster** icon .
- b. Click **Unregister** in the **Unregister cluster** dialog box.

A message is displayed that the remote cluster is unregistered successfully.

Results

You have unregistered the remote cluster from a team space on Rational® Test Automation Server. You cannot run test assets or resources on this remote cluster if it is added to your project. You might have to add it again to your team space, if you want to use the remote cluster in your projects.

Related information

[Management of remote Kubernetes clusters on page 234](#)

[Editing configurations of a remote Kubernetes cluster on page 244](#)

[Adding a remote Kubernetes cluster to a project on page 242](#)

[Removing a remote Kubernetes cluster from a project on page 246](#)

Test run configurations

You can configure and run tests in IBM® Rational® Test Automation Server after you add the test resources to your project in your team space.


Before you configure a test run, you must have completed the following tasks:

- Created a project on Rational® Test Automation Server or you must have been granted access to a project with the *Tester* role assigned. See [Test assets and a server project on page 616](#).
- Created tests in the desktop clients and committed the test assets and test resources to a remote repository. You must have added the remote repository to the project.
- Read the considerations you must take into account for certain test types. See [Prerequisites to running tests on page 198](#).

You must be a project *Owner* or a member with the *Tester* role assigned to configure and run a test.

You can find information about how to configure and run the following types of test assets:

Type of Suite or test	Suite or test	More information
AFT Suite	Suite	Configuring an AFT Suite run on page 255
API Suite	Suite	Configuring an API Suite or an API test run on page 267
Compound Test that contains HTML tests	Suite	Configuring a run of a Compound Test that contains HTML tests on page 280
Compound Test that contains Web UI tests	Suite	Configuring a run of a Compound Test that contains Web UI tests or a Functional Web UI test on page 291

Type of Suite or test	Suite or test	More information
Compound Test that contains mobile tests	Suite	Configuring a run of a Compound Test that contains mobile tests or a Functional Mobile test on page 304
Compound Test that contains performance tests	Suite	Configuring a run of a Compound Test that contains performance tests or a Performance test on page 331
Rate Schedule	Suite	Configuring a run of a Rate Schedule or VU Schedule on page 371
VU Schedule	Suite	Configuring a run of a Rate Schedule or VU Schedule on page 371
API test	Test	Configuring an API Suite or an API test run on page 267
Functional test  Note: The Functional test can contain any or the following tests: <ul style="list-style-type: none"> • Mobile tests • Web UI tests • Windows tests 	Test	<ul style="list-style-type: none"> • Configuring a run of a Compound Test that contains mobile tests or a Functional Mobile test on page 304 • Configuring a run of a Compound Test that contains Web UI tests or a Functional Web UI test on page 291 • Configuring a run of a Functional Windows test on page 319
Performance test	Test	Configuring a run of a Compound Test that contains performance tests or a Performance test on page 331

Configuring a Suite or test for a quick run

You can configure any type of Suite or test to be run on IBM® Rational® Test Automation Server when you want to quickly ensure that the test runs correctly. You might not want to set the different options for the test nor want to schedule the run.

Before you begin

You must have completed the following tasks:

- Read and completed the tasks mentioned in [Prerequisites to running tests on page 198](#) if they apply to the test you want to configure for a run.
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created a project in your team space in Rational® Test Automation Server. See [Adding a project on page 619](#).
- Added the remote Git repository that contains the test resources to the project. See [Adding repositories to a server project on page 620](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the test assets, and then click **Execution**.
3. Select the branch of the repository that contains the test assets.

The test assets that are contained in the selected branch of the repository are displayed in the following tabs on the **Execution** page:

Tab	Description
SUITES	Lists all suites, compound tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules that are in the selected branch of the repository.
TESTS	Lists all API tests, functional tests, or performance tests that are in the selected branch of the repository.
ADVANCED	Lists all assets that are displayed when custom filters are applied for assets in the selected branch of the repository.

4. Select the tab based on the type of test assets that you want to run as indicated in the following table:


If...	Then...
You want to run AFT Suites, API Suites, Compound Tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules,.	Click the SUITES tab.
You want to run API tests, functional tests, or performance tests.	Click the TESTS tab.
You want to find Suites and tests by using custom filters, and then run Suites and tests.	Click the ADVANCED tab.





































The test assets in the selected tab are displayed.

5. Identify the test asset that you want to run by performing any of the following actions:


- Scroll through the list.

 **Tip:** You can hover over the icon in the **Type** column to know the type of the test asset.


 **Note:** You can also identify the type of the asset from the icon that represents the test type as shown in the following table:

Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the AD-VANCED tab
	API test			
	Functional test			
	Performance test			
	AFT Suite			
	API Suite			
	Compound Test			
	HCL AppScan CodeSweep			
	JMeter Test			
	JUnit Test			
	Postman test			
	Rate Schedule			
	VU Schedule			

- Search for the test asset by entering any text contained in the test asset name in the **Search** field.

- Click the **Filter** icon  in the **SUITES** or **TESTS** tab to filter the displayed assets based on the asset type.

For example, select **API Suite** in the **SUITES** tab to display all API Suites or select **Functional Test** in the **TESTS** tab to display all functional tests that are in the selected branch of the repository.

- Click the **Filter** icon  in the **ADVANCED** tab, and then create a filter query by using the **New filter** option by performing the following steps:
 - a. Click **New filter**.
 - b. Enter a name for the filter.
 - c. Select an operator, and add a rule or a group of rules.
 - d. Add or enter the relevant parameters and either select or enter the condition and the criteria for the condition.

You can select a parameter from the following list:

- Type
 - Test Asset Name
 - Test Asset Path
 - Last Result
 - Next Run
 - Components
- e. Save the filter query to save and apply the filter query to filter the assets based on the query.

The test assets that match the filter criteria are displayed.

- Retrieve and apply a saved filter query, if you have saved filter queries previously by performing the following steps:



Note: The filter query applied previously is selected and the assets based on that filter query are displayed. To apply a different filter query, you must have created and saved the filter query.

- a. Click the **Filter** icon  in the **ADVANCED** tab.

The filter queries that you created and saved are displayed.


- b. Click the filter that you want to apply.

The test assets that match the filter criteria are displayed.

You have identified the test asset that you want to run.

Clicking the test name displays the **Details** panel. You can view the details of the test such as the description, the branch in the repository that contains the asset, the Git details, and the details of the commits to the

repository. You can also view the history of the test runs for the specific test under the **History** tab of the **Details** panel.

- Click the **Execute** icon  in the row of the identified test asset.

The **Execute test asset** dialog is displayed.



Notes:

- If you have configured some or all of the settings for the current test run, and you do not want to continue with those settings, you can reset the settings by clicking **Reset**.
- If you want to repeat a test run and do not want to use the saved settings from a previous run, you can reset all the saved settings to their default values by clicking **Reset**.

- Select the version of the test resources that you want to run, if you want to run a different version other than the latest version.
- No action is required if you want to initiate the test run immediately after you click **Execute**.
- Enter a label for the test run that helps you to identify the test on the **Results** page.

After the test run completes, the label that you entered is displayed for the test under the **Labels** column on the **Results** page.

- Click **Execute**.


The test run is initiated.

Results

You have started a test run.

What to do next

You can choose to perform any of the following tasks:

- View all the states of the test asset by clicking the **Show in the Progress page** icon  for the test asset for which you started or scheduled the run. See [Viewing the state of test assets on page 394](#).
- View the progress of the test from the **Progress** page. See [Viewing the progress of running test assets on page 398](#).
- View the results, reports, and logs of the test from the **Results** page after the test completes the run. See [Test results on page 455](#).

Related information

[Resetting the configuration settings for a test run on page 402](#)

[Viewing the progress of running test assets on page 398](#)

[Stopping test runs on page 403](#)

[Canceling scheduled test runs on page 406](#)

Configuring an AFT Suite run

After you added the test resources that you created in the desktop client to the project, you can configure an AFT Suite to be run on IBM® Rational® Test Automation Server or a remote Kubernetes cluster.

Before you begin

You must have completed the following tasks:

- Read and completed the tasks mentioned in [Prerequisites to running tests on page 198](#), if they apply to the test that you want to configure for a run.
- You must have read [Test run considerations for AFT Suites or Functional tests on page 199](#), if you want to configure a run for an AFT Suite that has the agent location configured in the AFT XML file.
- Read and completed the tasks mentioned in [Test run considerations for running tests on remote performance agents on page 211](#), if you want to use a remote agent as the location to run the test.
- Read and completed the tasks mentioned in [Considerations for using a remote Kubernetes cluster on page 235](#), if you want to use a remote Kubernetes cluster as a location to run the test.
- Completed the Xray integration, if you want to export the test results to a Jira issue in your Xray project in Jira. See [Integration with Xray test management for Jira on page 593](#).
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created a project in your team space in Rational® Test Automation Server. See [Adding a project on page 619](#).
- Added the remote Git repository that contains the test resources to the project. See [Adding repositories to a server project on page 620](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the test assets, and then click **Execution**.
3. Select the branch of the repository that contains the test assets.

The test assets that are contained in the selected branch of the repository are displayed in the following tabs on the **Execution** page:

Tab	Description
SUITES	Lists all suites, compound tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules that are in the selected branch of the repository.

Tab	Description
TESTS	Lists all API tests, functional tests, or performance tests that are in the selected branch of the repository.
ADVANCED	Lists all assets that are displayed when custom filters are applied for assets in the selected branch of the repository.

4. Select the tab based on the type of test assets that you want to run as indicated in the following table:


If...	Then...
You want to run AFT Suites, API Suites, Compound Tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules,.	Click the SUITES tab.
You want to run API tests, functional tests, or performance tests.	Click the TESTS tab.
You want to find Suites and tests by using custom filters, and then run Suites and tests.	Click the ADVANCED tab.










The test assets in the selected tab are displayed.


5. Identify the test asset that you want to run by performing any of the following actions:




























- Scroll through the list.


 **Tip:** You can hover over the icon in the **Type** column to know the type of the test asset.

 **Note:** You can also identify the type of the asset from the icon that represents the test type as shown in the following table:


Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the ADVANCED tab
	API test			
	Functional test			
	Performance test			



Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the ADVANCED tab
	AFT Suite			
	API Suite			
	Compound Test			
	HCL AppScan CodeSweep			
	JMeter Test			
	JUnit Test			
	Postman test			
	Rate Schedule			
	VU Schedule			

- Search for the test asset by entering any text contained in the test asset name in the **Search** field.
- Click the **Filter** icon  in the **SUITES** or **TESTS** tab to filter the displayed assets based on the asset type.

For example, select **API Suite** in the **SUITES** tab to display all API Suites or select **Functional Test** in the **TESTS** tab to display all functional tests that are in the selected branch of the repository.

- Click the **Filter** icon  in the **ADVANCED** tab, and then create a filter query by using the **New filter** option by performing the following steps:
 - Click **New filter**.
 - Enter a name for the filter.
 - Select an operator, and add a rule or a group of rules.
 - Add or enter the relevant parameters and either select or enter the condition and the criteria for the condition.

You can select a parameter from the following list:

- Type
- Test Asset Name

- Test Asset Path
 - Last Result
 - Next Run
 - Components
- e. Save the filter query to save and apply the filter query to filter the assets based on the query.

The test assets that match the filter criteria are displayed.

- Retrieve and apply a saved filter query, if you have saved filter queries previously by performing the following steps:



Note: The filter query applied previously is selected and the assets based on that filter query are displayed. To apply a different filter query, you must have created and saved the filter query.

- a. Click the **Filter** icon  in the **ADVANCED** tab.


The filter queries that you created and saved are displayed.

- b. Click the filter that you want to apply.

The test assets that match the filter criteria are displayed.

You have identified the test asset that you want to run.

Clicking the test name displays the **Details** panel. You can view the details of the test such as the description, the branch in the repository that contains the asset, the Git details, and the details of the commits to the repository. You can also view the history of the test runs for the specific test under the **History** tab of the **Details** panel.

6. Click the **Execute** icon  in the row of the identified test asset.

The **Execute test asset** dialog is displayed.



Notes:

- If you have configured some or all of the settings for the current test run, and you do not want to continue with those settings, you can reset the settings by clicking **Reset**.
- If you want to repeat a test run and do not want to use the saved settings from a previous run, you can reset all the saved settings to their default values by clicking **Reset**.

7. Select the version of the test resources that you want to run by completing any of the following actions:



Note: The test resources in the version can contain the test assets, datasets, AFT XML files, API environment tags, and other resources specific to projects created from any of the desktop clients.

- Expand the list in the **Version** field, find the version of the test resources, and then select the version.

Use the following details about the version of the test resources that are displayed to identify the version that you want:

- Commit message.
- Tags labeled by the user for the version committed.
- The user who committed the version to the repository.
- Relative time of the commit. For example, *2 hours ago* or *3 days ago*.

The list displays the versions of the test resources committed by all users to the branch in the repository. The versions are arranged with the latest version that is committed, and then followed by the versions committed previously.

- Expand the list in the **Version** field, and then search for the version that you want to select by entering a partial or the complete commit message of that version.

The version that matches the search criteria is displayed and it is selected for the test run.

The default value for the version selected for the run is the latest version in the selected branch of the repository. If you do not select any version, then the latest version is selected for the test run.



Notes:

- If you selected a version but you do not want to use that version in the test run, you can remove the selected version by clicking the ✕ icon. As a result, the default version is selected for the test run.
- If you repeated a test or ran the test again from the **Results** page, then the version of the test resources that you chose for the earlier run is shown as selected. You can either retain this version or select any other version from the list. You can also remove the previous version by clicking the ✕ icon.

8. Select the time for scheduling the test run from the following options:

- No action is required if you want to initiate the test run immediately after you click **Execute**.







Important: Click **Execute** only after you have configured the other settings in this dialog.



Note: The default time for scheduling a run is **Now**.

- Select **Schedule** and perform the following actions if you want to schedule a single test run or configure recurring test runs:


Schedule	Actions
<p>A single test run</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Select the Never option in the Repeat every field. Click Save. <p>The test run is scheduled to run at the selected time on the scheduled date.</p>
<p>Recurring test runs</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Set the frequency at which the test runs must run by entering the number in the Repeat every field, and then select the period from the list. You can select from the following options: <ul style="list-style-type: none"> ▪ Minute ▪ Hour ▪ Day ▪ Week ▪ Month <p>For example, if you want the test run to be run every day at the set time, enter 1 and, then select the Day(s) option.</p> <p>You can also schedule the test to run at the set time on specific weekdays by selecting the Week(s) option. For example, the following image displays the days selected for the test runs as <i>Monday, Wednesday, and Friday</i>:</p> <div data-bbox="708 1484 1421 1644" style="border: 1px solid black; padding: 5px; margin: 10px 0;"> <p>Repeat every <input style="width: 60px;" type="text" value="1"/> <input style="width: 100px;" type="text" value="Week(s)"/> </p> <p style="text-align: center;"> S M T W T F S </p> </div> Set the option to end the recurring test runs from the following options:

Schedule	Actions
	<ul style="list-style-type: none"> ▪ Select the Never option, if you do not want the recurring test runs to end. ▪ Select the On option, and then click the Calendar icon . You can select the date and time after which the scheduled test runs do not run. <p>f. Click Save.</p> <p>The recurring test runs are scheduled to start the first run at the selected time on the scheduled date, and to repeat the run at the frequency that you set.</p>

9. Enter a label for the test run that helps you to identify the test on the **Results** page.

After the test run completes, the label that you entered is displayed for the test under the **Labels** column on the **Results** page. After you have created a label, any member of the project can use that label.

The default value for the **Label** field is null or an empty field.

 **Important:** The configuration that you set for the test run in the **Execute test asset** dialog is preserved when you run the same test again. Those changes are not visible when another user logs in to Rational® Test Automation Server. For example, if you created new variables on the server, those variables are available only for you when the same test is run again.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

If you are running an AFT Suite and the following conditions are true, then you must perform the next step:

- You do not want to run the test on the agent configured in the AFT XML file.
- You have not selected the agent from the **Override** column in the **Location** tab.

10. Click **Advanced** to make the following advanced configurations:

- a. Add the following setting in the **Program Arguments** field:

```
-swaplocation <configured_agent_location>:<overriding_agent_location>
```

For example, if the configured agent location is *91.2.352.24* and the remote agent where you want to run the test is *100.35.117.164*, then the entry in the **Program Arguments** field is as follows:

```
-swaplocation 91.2.352.24:100.35.117.164
```

If the test that you want to configure supports the presentation of the test results data in different formats, then do the required tasks in the following scenarios:

If...	Then...
You want the test results data in different supported formats.	Enter <code>-history cisterna</code> in the Program Arguments field.
You want to obtain the test results as a Jaeger trace.	Enter <code>-history jaeger</code> in the Program Arguments field.
You want the report as a test log and as a Jaeger trace.	Enter <code>-history jaeger, testlog</code> in the Program Arguments field.
Jaeger is already set as a program argument and you want to remove the Jaeger trace for your test.	Delete the <code>-history jaeger</code> entry in the Program Arguments field.



Note: The default report format is the *test log* format for the test reports.



Note: You must separate the arguments or variables with a white space when you enter them in the same line or start each argument or variable on a new line.

The default value for each of the fields for the advanced settings is null or an empty field.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

11. Follow the instructions if you are running a test asset that contains datasets:

- a. Click the **DATA SOURCES** tab, if it is not already open.
- b. Consider the following information about datasets before you select a dataset:

The default value for the datasets in the **DATA SOURCES** tab is null if the test asset did not have an associated dataset. If the asset had an associated dataset, the default value is the associated dataset.

You can utilize the dataset stored as an Excel or CSV file to override the original dataset associated with the Suite, test, or schedule. For example, when you have associated a dataset in `.xlsx`, `.xls`, or `.csv` format with the test or schedule in desktop clients and if you have another set of data stored in an Excel or CSV file, then you can select that dataset from the **Override** list.



Remember: You must have uploaded the dataset as an Excel or CSV file into the Git repository, and ensured that both the original dataset (from the test asset) and new datasets (added to the project) have the same column names.

c. Select the dataset that you want to use in the test run from any of the following options:

- Select the dataset that is displayed as the default dataset when the test asset contains a single dataset.



Note: If there is only one dataset in the test asset, then that dataset is displayed as the default dataset.

- Select the dataset from the list.



Note: If there are multiple datasets in the test asset, the datasets are listed in their increasing alphabetical order.

- Select the dataset from the **Override** list to override the dataset that was associated with the test in the desktop client.





Important: If the test contains an encrypted dataset, the Project Owner must classify it in the **DATA SECURITY** tab on the Project page before you can select it. You must have added datasets to your project from the **Dataset** page for the datasets to be displayed in the **Override** list.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

12. Follow the instructions if the test requires a variable that must be passed to the test at the test run time.

a. Click the **VARIABLES** tab, if it is not already open.

b. Choose one of the following methods to add the variables:

- To add new variables manually, click the **Add Variable** icon , enter the name, and value of the variable.
- To add new variables from your local computer or from the Git repository that is associated with your server project, click the **Upload** icon  and select the **Upload from local system** or **Browse from server** to select the variable file.



Note: You must have created a file with the variables before you can select the file.


The default value for the variables is null or an empty field.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

13. Follow the instructions if you want to export the test results to a Jira issue in your Xray project in Jira:

a. Click the **RESULTS** tab.

b. Perform the actions listed in the following table to select an option to export the test results:

Export options	Results in...	Actions
Jira Xray	The test results of the test that you configure for a run are exported to the selected issue after the test run is completed. The results are exported to the selected issue that exists in your Xray project in Jira.	<p>Perform the following steps:</p> <ul style="list-style-type: none"> i. Select the Jira Xray option. <p>The Select issue field is displayed.</p> <ul style="list-style-type: none"> ii. Click the field to view the list of issues in the Xray project in Jira. Alternatively, you can enter the issue key in the Jira Xray project. iii. Select the issue to which you want to export the test results. iv. Continue with the next step.
New Jira item	The test results of the test that you configure for a run are exported to a new Jira issue. The issue is created in your project in Jira after the test run is completed.	<p>Perform the following steps:</p> <ul style="list-style-type: none"> i. Select the New Jira item option. ii. Continue with the next step.
None	<p>The test results are not exported.</p> <p> Note: This is the default option.</p>	<p>Perform the following steps when:</p> <ul style="list-style-type: none"> ▪ You do not want to export the test results. ▪ You selected a Jira issue in a previous run of the same test, and you do not want to export results of the current run. <ul style="list-style-type: none"> i. Select the None option. ii. Continue with the next step.

14. Select any of the following steps that depends on the location to run the test:

- If you want to run the tests on the remote agents, go to [Step 15 on page 264](#).
- If you want to run the tests on a Kubernetes cluster, go to [Step 16 on page 266](#).

15. Follow the instructions if you want to run a test on the remote performance agent that is connected either to the default Kubernetes cluster or the remote Kubernetes cluster:

- a. Click the **LOCATION** tab, if it is not already open.

The performance agents that are configured in the test asset are listed under the **Host** column. The information about the availability of the performance agent is displayed.



Note: You must have added performance agents to your project from the **Agents and Intercepts** page for the performance agents to be displayed under the **Override** column.

The default value for the performance agents is null or an empty field if no performance agents were configured in the test asset. If the test asset contains performance agents that are configured, then the default performance agent is the first item to be displayed on the list of performance agents listed in the increasing alphabetical order.

- b. Select the performance agent where you want to run the test asset in the following scenarios:

If...	Then...	Action
The performance agent that is specified in the test assets is available and no other performance agents are added to the project.	The performance agent that is specified in the test assets is displayed in the Override column.	Select the performance agent in the Host that is also displayed in the Override column as the location to run the test.
The performance agent that is specified in the test assets is not available and no other performance agents are added to the project.	There is no performance agent displayed for override in the Override column.	You cannot run the test on the performance agent. You must wait until the performance agent is available to run the test on the agent.
The agent that is specified in the test assets is available and there are other agents that are added to the project.	Performance agents that have capabilities that match with the agent capabilities specified in the test assets are listed in the Override column as follows: <ul style="list-style-type: none"> ▪ Performance agents with capabilities that best match the capabilities in the agent in the asset are at the top of the list. ▪ Performance agents with capabilities that do not match with the ca- 	Perform any of the following actions: <ul style="list-style-type: none"> ▪ Select the performance agent in the Host that is also displayed in the Override column as the location to run the test. ▪ Select a performance agent in the Override column as the location to run the test.

If...	Then...	Action
	<p>pabilities in the agent in the asset are listed subsequently.</p> <p>You can find details of both performance agents for the matching capabilities when you hover over the performance agent in the Override column.</p>	

c. Click **Execute**.

The test run is initiated.

16. Follow the instructions if you want to change the location of a Kubernetes cluster for running the test:

a. Click the **LOCATION** tab, if it is not already open.

The **Default Cluster** is the default location where the test runs, and it is listed under the **Host** column. The information about the availability of the default location is displayed.



Important: You must have added the following remote hosts to your project that are then displayed under the **Override** column:

- Docker hosts that are registered with Rational® Test Automation Server.
- Kubernetes clusters that are registered with a team space.



Notes:

- If the remote Docker hosts or clusters are not added to your project, the option **No override options** is displayed as the default value and the test runs in the Kubernetes cluster of Rational® Test Automation Server.
- If remote Docker hosts or clusters are added to your project, the added Docker hosts or clusters are displayed along with their availability status and ownership information.

b. Select the location where you want to run the test asset from the following options:

- Select the **Default Cluster** when no remote Docker hosts or clusters are available in your project.
- Select the remote Docker host or cluster from the list when a remote Docker host or cluster is available in your project.
- Select **No override options**, if you selected any remote Docker host or cluster and want to revert to the **Default Cluster** to run the test asset.

c. Click **Execute**.


The test run is initiated.

Results

You have started or scheduled a test run of an AFT Suite.

What to do next

You can choose to perform any of the following tasks:

- Stop the test run at any point after the test run is initiated from the **Execution** page. See [Stopping test runs on page 403](#).
- Cancel a scheduled test run from the **Execution** page. See [Canceling scheduled test runs on page 406](#).
- View all the states of the test asset by clicking the **Show in the Progress page** icon  for the test asset for which you started or scheduled the run. See [Viewing the state of test assets on page 394](#).
- View the progress of the test from the **Progress** page. See [Viewing the progress of running test assets on page 398](#).
- View the results, reports, and logs of the test from the **Results** page after the test completes the run. See [Test results on page 455](#).

Related information

[Resetting the configuration settings for a test run on page 402](#)

[Test run configurations on page 249](#)

Configuring an API Suite or an API test run

After you added the test resources that you created in the desktop client to the project, you can configure an API Suite or an API test to be run on IBM® Rational® Test Automation Server or a remote Kubernetes cluster.

Before you begin

You must have completed the following tasks:

- Read and completed the tasks mentioned in [Prerequisites to running tests on page 198](#), if they apply to the test that you want to configure for a run.
- Read and completed the tasks mentioned in [Considerations for using a remote Kubernetes cluster on page 235](#), if you want to use a remote Kubernetes cluster as a location to run the test.
- Completed the following tasks if you are running API Suites or tests that use a transport and the transport requires third-party application `Jar` files for a successful run:

- Read [Test run considerations for API Suites or API tests on page 200](#).
- Copied the third-party application `JAR` files to the third-party application folder on the computer where Rational® Test Automation Server is installed, if you are running the API Suite in Kubernetes. See [Copying third-party application Jars to Kubernetes on page 118](#).
- Copied the third-party application `JAR` files to the third-party application folder on the computer where the remote Docker host is installed, if you are running the API Suite on a remote Docker host. See [Copying third-party application Jars to a remote Docker host on page 223](#).
- Completed the Xray integration, if you want to export the test results to a Jira issue in your Xray project in Jira. See [Integration with Xray test management for Jira on page 593](#).
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created a project in your team space in Rational® Test Automation Server. See [Adding a project on page 619](#).
- Added the remote Git repository that contains the test resources to the project. See [Adding repositories to a server project on page 620](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the test assets, and then click **Execution**.
3. Select the branch of the repository that contains the test assets.

The test assets that are contained in the selected branch of the repository are displayed in the following tabs on the **Execution** page:

Tab	Description
SUITES	Lists all suites, compound tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules that are in the selected branch of the repository.
TESTS	Lists all API tests, functional tests, or performance tests that are in the selected branch of the repository.
ADVANCED	Lists all assets that are displayed when custom filters are applied for assets in the selected branch of the repository.

4. Select the tab based on the type of test assets that you want to run as indicated in the following table:


If...	Then...
You want to run AFT Suites, API Suites, Compound Tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules,.	Click the SUITES tab.
You want to run API tests, functional tests, or performance tests.	Click the TESTS tab.
You want to find Suites and tests by using custom filters, and then run Suites and tests.	Click the ADVANCED tab.






















The test assets in the selected tab are displayed.


5. Identify the test asset that you want to run by performing any of the following actions:
















- Scroll through the list.


 **Tip:** You can hover over the icon in the **Type** column to know the type of the test asset.

 **Note:** You can also identify the type of the asset from the icon that represents the test type as shown in the following table:


Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the ADVANCED tab
	API test			
	Functional test			
	Performance test			
	AFT Suite			
	API Suite			
	Compound Test			
	HCL AppScan CodeSweep			



Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the AD-VANCED tab
	JMeter Test			
	JUnit Test			
	Postman test			
	Rate Schedule			
	VU Schedule			

- Search for the test asset by entering any text contained in the test asset name in the **Search** field.
- Click the **Filter** icon  in the **SUITES** or **TESTS** tab to filter the displayed assets based on the asset type.

For example, select **API Suite** in the **SUITES** tab to display all API Suites or select **Functional Test** in the **TESTS** tab to display all functional tests that are in the selected branch of the repository.

- Click the **Filter** icon  in the **ADVANCED** tab, and then create a filter query by using the **New filter** option by performing the following steps:
 - Click **New filter**.
 - Enter a name for the filter.
 - Select an operator, and add a rule or a group of rules.
 - Add or enter the relevant parameters and either select or enter the condition and the criteria for the condition.

You can select a parameter from the following list:

- Type
 - Test Asset Name
 - Test Asset Path
 - Last Result
 - Next Run
 - Components
- Save the filter query to save and apply the filter query to filter the assets based on the query.

The test assets that match the filter criteria are displayed.

- Retrieve and apply a saved filter query, if you have saved filter queries previously by performing the following steps:



Note: The filter query applied previously is selected and the assets based on that filter query are displayed. To apply a different filter query, you must have created and saved the filter query.

- Click the **Filter** icon  in the **ADVANCED** tab.


The filter queries that you created and saved are displayed.

- Click the filter that you want to apply.

The test assets that match the filter criteria are displayed.

You have identified the test asset that you want to run.

Clicking the test name displays the **Details** panel. You can view the details of the test such as the description, the branch in the repository that contains the asset, the Git details, and the details of the commits to the repository. You can also view the history of the test runs for the specific test under the **History** tab of the **Details** panel.

- Click the **Execute** icon  in the row of the identified test asset.

The **Execute test asset** dialog is displayed.



Notes:

- If you have configured some or all of the settings for the current test run, and you do not want to continue with those settings, you can reset the settings by clicking **Reset**.
- If you want to repeat a test run and do not want to use the saved settings from a previous run, you can reset all the saved settings to their default values by clicking **Reset**.

- Select the version of the test resources that you want to run by completing any of the following actions:



Note: The test resources in the version can contain the test assets, datasets, AFT XML files, API environment tags, and other resources specific to projects created from any of the desktop clients.

- Expand the list in the **Version** field, find the version of the test resources, and then select the version.

Use the following details about the version of the test resources that are displayed to identify the version that you want:

- Commit message.
- Tags labeled by the user for the version committed.

- The user who committed the version to the repository.
- Relative time of the commit. For example, *2 hours ago* or *3 days ago*.

The list displays the versions of the test resources committed by all users to the branch in the repository. The versions are arranged with the latest version that is committed, and then followed by the versions committed previously.

- Expand the list in the **Version** field, and then search for the version that you want to select by entering a partial or the complete commit message of that version.

The version that matches the search criteria is displayed and it is selected for the test run.

The default value for the version selected for the run is the latest version in the selected branch of the repository. If you do not select any version, then the latest version is selected for the test run.



Notes:

- If you selected a version but you do not want to use that version in the test run, you can remove the selected version by clicking the **✕** icon. As a result, the default version is selected for the test run.
- If you repeated a test or ran the test again from the **Results** page, then the version of the test resources that you chose for the earlier run is shown as selected. You can either retain this version or select any other version from the list. You can also remove the previous version by clicking the **✕** icon.

8. Select the time for scheduling the test run from the following options:

- No action is required if you want to initiate the test run immediately after you click **Execute**.






Important: Click **Execute** only after you have configured the other settings in this dialog.



Note: The default time for scheduling a run is **Now**.

- Select **Schedule** and perform the following actions if you want to schedule a single test run or configure recurring test runs:


Schedule	Actions
A single test run	Perform the following steps: <ol style="list-style-type: none"> a. Click the Calendar icon  in the row of the Start field. b. Select the date. c. Select the time at which the test run must start. d. Select the Never option in the Repeat every field. e. Click Save.

Schedule	Actions
	The test run is scheduled to run at the selected time on the scheduled date.
Recurring test runs	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Set the frequency at which the test runs must run by entering the number in the Repeat every field, and then select the period from the list. You can select from the following options: <ul style="list-style-type: none"> Minute Hour Day Week Month <p>For example, if you want the test run to be run every day at the set time, enter 1 and, then select the Day(s) option.</p> <p>You can also schedule the test to run at the set time on specific weekdays by selecting the Week(s) option. For example, the following image displays the days selected for the test runs as <i>Monday, Wednesday, and Friday</i>:</p> <div data-bbox="708 1178 1421 1339" data-label="Image"> </div> Set the option to end the recurring test runs from the following options: <ul style="list-style-type: none"> Select the Never option, if you do not want the recurring test runs to end. Select the On option, and then click the Calendar icon . You can select the date and time after which the scheduled test runs do not run. Click Save. <p>The recurring test runs are scheduled to start the first run at the selected time on the scheduled date, and to repeat the run at the frequency that you set.</p>

9. Enter a label for the test run that helps you to identify the test on the **Results** page.

After the test run completes, the label that you entered is displayed for the test under the **Labels** column on the **Results** page. After you have created a label, any member of the project can use that label.

The default value for the **Label** field is null or an empty field.

 **Important:** The configuration that you set for the test run in the **Execute test asset** dialog is preserved when you run the same test again. Those changes are not visible when another user logs in to Rational® Test Automation Server. For example, if you created new variables on the server, those variables are available only for you when the same test is run again.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

10. Click **Advanced** to make the following advanced configurations:

a. Enter any JVM arguments that must be passed to the test run at run time in the **JVM Arguments** field, if applicable for the test.

For example, you can set a maximum Java heap size.

b. Enter program arguments that must be passed to the test run at run time in the **Program Arguments**, if applicable for the test.

Perform the following actions, if applicable for your test:

- Configuring Jaeger tracing.

If the test that you want to configure supports the presentation of the test results data in different formats, then do the required tasks in the following scenarios:

If...	Then...
You want the test results data in different supported formats.	Enter <code>-history cisterna</code> in the Program Arguments field.
You want to obtain the test results as a Jaeger trace.	Enter <code>-history jaeger</code> in the Program Arguments field.
You want the report as a test log and as a Jaeger trace.	Enter <code>-history jaeger, testlog</code> in the Program Arguments field.
Jaeger is already set as a program argument and you want to remove the Jaeger trace for your test.	Delete the <code>-history jaeger</code> entry in the Program Arguments field.




Note: The default report format is the *test log* format for the test reports.

- Overriding results database that is configured in the API Suite project.



Note: You can use any of the databases that are specified in the system requirements as supported databases in Rational® Integration Tester.

Enter the following arguments in the **Program Arguments** field, if you want to override the **Results Database**:

Program Arguments	Description
-resultDatabaseURL <database_URL>	Specifies the URL of the results database that overrides the results database that is configured for the project in Rational® Integration Tester.
-resultDatabaseDriver <driver_name>	Specifies the driver of the results database that overrides the results database that is configured for the project in Rational® Integration Tester. For example, the driver for a MySQL database is <i>com.mysql-cj.jdbc.Driver</i> .
-resultDatabaseUser- name <user_name>	Specifies the username that is authorized to access the results database that overrides the results database that is configured for the project in Rational® Integration Tester.
-resultDatabasePa- ssword <encrypted_ password>	Specifies the password that is to be used to authenticate the user to access the results database that overrides the results database that is configured for the project in Rational® Integration Tester.  Note: You can encrypt the password by using the Encrypt-Password application that is available in the installation directory of Rational® Integration Tester.

For example, if you want to override the results database that is configured in the Rational® Integration Tester project with a different results database that has the following details:

- MySQL database as the results database with the URL as *jdbc:mysql://localhost:3306/myresultsdb*
- *com.mysql.cj.jdbc.Driver* as the driver for the MySQL database
- *tester* as the username to access the MySQL database
- *#com.ghc.1!9a8521137E0A12994ACF4434E315A2B9* as the encrypted password to authenticate the user name *tester*

The entry in the **Program Arguments** field must be as follows:

```
-resultDatabaseURL jdbc:mysql://localhost:3306/myresultsdb
-resultDatabaseDriver com.mysql.cj.jdbc.Driver
-resultDatabaseUsername tester
-resultDatabasePassword #com.ghc.1!9a8521137E0A12994ACF4434E315A2B9
```



Note: You must separate entries in the field with white spaces or with a new-line character.

11. Follow the instructions if the API Suite has an environment or secrets configured:
 - a. Click the **ENVIRONMENT** tab, if it is not already open.
 - b. Select the API test environment from the list if there are multiple environments configured in the test asset.
 - c. Select the secrets collection that contains the secrets to be used for the test run.



Notes:

- The test asset that was created in the desktop client and added to the Git repository must have the environments defined as part of the API test project.
- If the test asset contained secrets, then you must create those secrets in secrets collections in the project.

The default value for the environment is the environment configured in the test asset. The default value for secrets is null or an empty field.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

12. Follow the instructions if you are running a test asset that contains datasets:

- a. Click the **DATA SOURCES** tab, if it is not already open.
- b. Consider the following information about datasets before you select a dataset:

The default value for the datasets in the **DATA SOURCES** tab is null if the test asset did not have an associated dataset. If the asset had an associated dataset, the default value is the associated dataset.

You can utilize the dataset stored as an Excel or CSV file to override the original dataset associated with the Suite, test, or schedule. For example, when you have associated a dataset in `.xlsx`, `.xls`, or `.csv` format with the test or schedule in desktop clients and if you have another set of data stored in an Excel or CSV file, then you can select that dataset from the **Override** list.



Remember: You must have uploaded the dataset as an Excel or CSV file into the Git repository, and ensured that both the original dataset (from the test asset) and new datasets (added to the project) have the same column names.

- c. Select the dataset that you want to use in the test run from any of the following options:

- Select the dataset that is displayed as the default dataset when the test asset contains a single dataset.



Note: If there is only one dataset in the test asset, then that dataset is displayed as the default dataset.

- Select the dataset from the list.



Note: If there are multiple datasets in the test asset, the datasets are listed in their increasing alphabetical order.

- Select the dataset from the **Override** list to override the dataset that was associated with the test in the desktop client.





Important: If the test contains an encrypted dataset, the Project Owner must classify it in the **DATA SECURITY** tab on the Project page before you can select it. You must have added datasets to your project from the **Dataset** page for the datasets to be displayed in the **Override** list.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

13. Follow the instructions if the test requires a variable that must be passed to the test at the test run time.

a. Click the **VARIABLES** tab, if it is not already open.

b. Choose one of the following methods to add the variables:

- To add new variables manually, click the **Add Variable** icon , enter the name, and value of the variable.
- To add new variables from your local computer or from the Git repository that is associated with your server project, click the **Upload** icon  and select the **Upload from local system** or **Browse from server** to select the variable file.



Note: You must have created a file with the variables before you can select the file.


The default value for the variables is null or an empty field.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

14. Follow the instructions if you want to export the test results to a Jira issue in your Xray project in Jira:

a. Click the **RESULTS** tab.


b. Perform the actions listed in the following table to select an option to export the test results:

Export options	Results in...	Actions
Jira Xray	<p>The test results of the test that you configure for a run are exported to the selected issue after the test run is completed. The results are exported to the selected issue that exists in your Xray project in Jira.</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> i. Select the Jira Xray option. <p>The Select issue field is displayed.</p> <ol style="list-style-type: none"> ii. Click the field to view the list of issues in the Xray project in Jira. Alternatively, you can enter the issue key in the Jira Xray project. iii. Select the issue to which you want to export the test results. iv. Continue with the next step.
New Jira item	<p>The test results of the test that you configure for a run are exported to a new Jira issue. The issue is created in your project in Jira after the test run is completed.</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> i. Select the New Jira item option. ii. Continue with the next step.
None	<p>The test results are not exported.</p> <p> Note: This is the default option.</p>	<p>Perform the following steps when:</p> <ul style="list-style-type: none"> ▪ You do not want to export the test results. ▪ You selected a Jira issue in a previous run of the same test, and you do not want to export results of the current run. <ol style="list-style-type: none"> i. Select the None option. ii. Continue with the next step.

15. Follow the instructions if you want to change the location of a Kubernetes cluster for running the test:

- a. Click the **LOCATION** tab, if it is not already open.

The **Default Cluster** is the default location where the test runs, and it is listed under the **Host** column. The information about the availability of the default location is displayed.

 **Important:** You must have added the following remote hosts to your project that are then displayed under the **Override** column:



- Docker hosts that are registered with Rational® Test Automation Server.
- Kubernetes clusters that are registered with a team space.

**Notes:**

- If the remote Docker hosts or clusters are not added to your project, the option **No override options** is displayed as the default value and the test runs in the Kubernetes cluster of Rational® Test Automation Server.
- If remote Docker hosts or clusters are added to your project, the added Docker hosts or clusters are displayed along with their availability status and ownership information.

b. Select the location where you want to run the test asset from the following options:

- Select the **Default Cluster** when no remote Docker hosts or clusters are available in your project.
- Select the remote Docker host or cluster from the list when a remote Docker host or cluster is available in your project.
- Select **No override options**, if you selected any remote Docker host or cluster and want to revert to the **Default Cluster** to run the test asset.

c. Click **Execute**.


The test run is initiated.

Results

You have started or scheduled a test run of an API Suite.

What to do next

You can choose to perform any of the following tasks:

- Stop the test run at any point after the test run is initiated from the **Execution** page. See [Stopping test runs on page 403](#).
- Cancel a scheduled test run from the **Execution** page. See [Canceling scheduled test runs on page 406](#).
- View all the states of the test asset by clicking the **Show in the Progress page** icon  for the test asset for which you started or scheduled the run. See [Viewing the state of test assets on page 394](#).
- View the progress of the test from the **Progress** page. See [Viewing the progress of running test assets on page 398](#).
- View the results, reports, and logs of the test from the **Results** page after the test completes the run. See [Test results on page 455](#).

Related information

[Resetting the configuration settings for a test run on page 402](#)


[Test run configurations on page 249](#)

Configuring a run of a Compound Test that contains HTML tests

After you added the test resources that you created in the desktop client to the project, you can configure a Compound Test that contains HTML tests to be run on IBM® Rational® Test Automation Server or a remote Kubernetes cluster.

Before you begin

You must have completed the following tasks:

- When you want to run a Compound Test, you must have authored a Compound Test in IBM® Rational® Functional Tester and must have completed the following tasks when you created the Compound test:
 - Created a Compound Test under a Web UI project in your workspace.
 - Added a Functional HTML Test under the Compound Test.
 - Used the **startBrowser(Firefox,<URL>)** attribute with the browser value set to *Firefox* in the Functional test script.
-  **Restriction:** Functional test scripts that contain the **startApp()** or **callScript()** attributes are treated as non-HTML scripts and are not supported to be used in test scripts in the HTML tests.
- Ran the Compound Test that contains the HTML tests on IBM® Rational® Functional Tester.
- Committed the projects that include the Web UI project (with the Compound Test) and the Functional test (with the HTML tests) to the remote repository.
- Read and completed the tasks mentioned in [Considerations for using a remote Kubernetes cluster on page 235](#), if you want to use a remote Kubernetes cluster as a location to run the test.
- Completed the Xray integration, if you want to export the test results to a Jira issue in your Xray project in Jira. See [Integration with Xray test management for Jira on page 593](#).
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created a project in your team space in Rational® Test Automation Server. See [Adding a project on page 619](#).
- Added the remote Git repository that contains the test resources to the project. See [Adding repositories to a server project on page 620](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the test assets, and then click **Execution**.
3. Select the branch of the repository that contains the test assets.

The test assets that are contained in the selected branch of the repository are displayed in the following tabs on the **Execution** page:

Tab	Description
SUITES	Lists all suites, compound tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules that are in the selected branch of the repository.
TESTS	Lists all API tests, functional tests, or performance tests that are in the selected branch of the repository.
ADVANCED	Lists all assets that are displayed when custom filters are applied for assets in the selected branch of the repository.

4. Select the tab based on the type of test assets that you want to run as indicated in the following table:

If...	Then...
You want to run AFT Suites, API Suites, Compound Tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules,.	Click the SUITES tab.
You want to run API tests, functional tests, or performance tests.	Click the TESTS tab.
You want to find Suites and tests by using custom filters, and then run Suites and tests.	Click the ADVANCED tab.

The test assets in the selected tab are displayed.


5. Identify the test asset that you want to run by performing any of the following actions:
 - Scroll through the list.








































Tip: You can hover over the icon in the **Type** column to know the type of the test asset.




Note: You can also identify the type of the asset from the icon that represents the test type as shown in the following table:



Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the ADVANCED tab
	API test			
	Functional test			
	Performance test			
	AFT Suite			
	API Suite			
	Compound Test			
	HCL AppScan CodeSweep			
	JMeter Test			
	JUnit Test			
	Postman test			
	Rate Schedule			
	VU Schedule			

- Search for the test asset by entering any text contained in the test asset name in the **Search** field.
- Click the **Filter** icon  in the **SUITES** or **TESTS** tab to filter the displayed assets based on the asset type.

For example, select **API Suite** in the **SUITES** tab to display all API Suites or select **Functional Test** in the **TESTS** tab to display all functional tests that are in the selected branch of the repository.

- Click the **Filter** icon  in the **ADVANCED** tab, and then create a filter query by using the **New filter** option by performing the following steps:
 - Click **New filter**.
 - Enter a name for the filter.

- c. Select an operator, and add a rule or a group of rules.
- d. Add or enter the relevant parameters and either select or enter the condition and the criteria for the condition.

You can select a parameter from the following list:

- Type
 - Test Asset Name
 - Test Asset Path
 - Last Result
 - Next Run
 - Components
- e. Save the filter query to save and apply the filter query to filter the assets based on the query.

The test assets that match the filter criteria are displayed.

- Retrieve and apply a saved filter query, if you have saved filter queries previously by performing the following steps:



Note: The filter query applied previously is selected and the assets based on that filter query are displayed. To apply a different filter query, you must have created and saved the filter query.

- a. Click the **Filter** icon  in the **ADVANCED** tab.


The filter queries that you created and saved are displayed.

- b. Click the filter that you want to apply.

The test assets that match the filter criteria are displayed.

You have identified the test asset that you want to run.

Clicking the test name displays the **Details** panel. You can view the details of the test such as the description, the branch in the repository that contains the asset, the Git details, and the details of the commits to the repository. You can also view the history of the test runs for the specific test under the **History** tab of the **Details** panel.

6. Click the **Execute** icon  in the row of the identified test asset.

The **Execute test asset** dialog is displayed.



Notes:



- If you have configured some or all of the settings for the current test run, and you do not want to continue with those settings, you can reset the settings by clicking **Reset**.
- If you want to repeat a test run and do not want to use the saved settings from a previous run, you can reset all the saved settings to their default values by clicking **Reset**.

7. Select the version of the test resources that you want to run by completing any of the following actions:



Note: The test resources in the version can contain the test assets, datasets, AFT XML files, API environment tags, and other resources specific to projects created from any of the desktop clients.

- Expand the list in the **Version** field, find the version of the test resources, and then select the version.

Use the following details about the version of the test resources that are displayed to identify the version that you want:

- Commit message.
- Tags labeled by the user for the version committed.
- The user who committed the version to the repository.
- Relative time of the commit. For example, *2 hours ago* or *3 days ago*.

The list displays the versions of the test resources committed by all users to the branch in the repository. The versions are arranged with the latest version that is committed, and then followed by the versions committed previously.

- Expand the list in the **Version** field, and then search for the version that you want to select by entering a partial or the complete commit message of that version.

The version that matches the search criteria is displayed and it is selected for the test run.

The default value for the version selected for the run is the latest version in the selected branch of the repository. If you do not select any version, then the latest version is selected for the test run.



Notes:

- If you selected a version but you do not want to use that version in the test run, you can remove the selected version by clicking the ✕ icon. As a result, the default version is selected for the test run.
- If you repeated a test or ran the test again from the **Results** page, then the version of the test resources that you chose for the earlier run is shown as selected. You can either retain this version or select any other version from the list. You can also remove the previous version by clicking the ✕ icon.

8. Select the time for scheduling the test run from the following options:

- No action is required if you want to initiate the test run immediately after you click **Execute**.






Important: Click **Execute** only after you have configured the other settings in this dialog.



Note: The default time for scheduling a run is **Now**.

- Select **Schedule** and perform the following actions if you want to schedule a single test run or configure recurring test runs:


Schedule	Actions
A single test run	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Select the Never option in the Repeat every field. Click Save. <p>The test run is scheduled to run at the selected time on the scheduled date.</p>
Recurring test runs	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Set the frequency at which the test runs must run by entering the number in the Repeat every field, and then select the period from the list. You can select from the following options: <ul style="list-style-type: none"> ▪ Minute ▪ Hour ▪ Day ▪ Week ▪ Month <p>For example, if you want the test run to be run every day at the set time, enter 1 and, then select the Day(s) option.</p> <p>You can also schedule the test to run at the set time on specific weekdays by selecting the Week(s) option. For example, the following image displays the days selected for the test runs as <i>Monday, Wednesday, and Friday</i>:</p>

Schedule	Actions
	<div data-bbox="711 262 1419 422" style="border: 1px solid black; padding: 5px; margin-bottom: 10px;"> Repeat every <input type="text" value="1"/> <input type="text" value="Week(s)"/> ▾ <div style="display: flex; justify-content: center; gap: 10px;"> S M T W T F S </div> </div> <p>e. Set the option to end the recurring test runs from the following options:</p> <ul style="list-style-type: none"> ▪ Select the Never option, if you do not want the recurring test runs to end. ▪ Select the On option, and then click the Calendar icon . You can select the date and time after which the scheduled test runs do not run. <p>f. Click Save.</p> <p>The recurring test runs are scheduled to start the first run at the selected time on the scheduled date, and to repeat the run at the frequency that you set.</p>

9. Enter a label for the test run that helps you to identify the test on the **Results** page.

After the test run completes, the label that you entered is displayed for the test under the **Labels** column on the **Results** page. After you have created a label, any member of the project can use that label.

The default value for the **Label** field is null or an empty field.

 **Important:** The configuration that you set for the test run in the **Execute test asset** dialog is preserved when you run the same test again. Those changes are not visible when another user logs in to Rational® Test Automation Server. For example, if you created new variables on the server, those variables are available only for you when the same test is run again.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

10. Click **Advanced** to make the following advanced configurations:

a. Enter any JVM arguments that must be passed to the test run at run time in the **JVM Arguments** field, if applicable for the test.

For example, you can set a maximum Java heap size.

b. Enter program arguments that must be passed to the test run at run time in the **Program Arguments**, if applicable for the test.

If the test that you want to configure supports the presentation of the test results data in different formats, then do the required tasks in the following scenarios:

If...	Then...
You want the test results data in different supported formats.	Enter <code>-history cisterna</code> in the Program Arguments field.
You want to obtain the test results as a Jaeger trace.	Enter <code>-history jaeger</code> in the Program Arguments field.
You want the report as a test log and as a Jaeger trace.	Enter <code>-history jaeger, testlog</code> in the Program Arguments field.
Jaeger is already set as a program argument and you want to remove the Jaeger trace for your test.	Delete the <code>-history jaeger</code> entry in the Program Arguments field.



Note: The default report format is the *test log* format for the test reports.

- c. Enter the environment variables that must be passed to the test run at run time in the **Environment Variables** field, if applicable for the test.

For example, enter the environment variables when the third-party libraries that are used in the test run refer to the environment variables for configuration.



Note: You must separate the arguments or variables with a white space when you enter them in the same line or start each argument or variable on a new line.

The default value for each of the fields for the advanced settings is null or an empty field.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

11. Follow the instructions if you are running a test asset that contains datasets:

- a. Click the **DATA SOURCES** tab, if it is not already open.
- b. Consider the following information about datasets before you select a dataset:

The default value for the datasets in the **DATA SOURCES** tab is null if the test asset did not have an associated dataset. If the asset had an associated dataset, the default value is the associated dataset.

You can utilize the dataset stored as an Excel or CSV file to override the original dataset associated with the Suite, test, or schedule. For example, when you have associated a dataset in `.xlsx`, `.xls`, or `.csv` format with the test or schedule in desktop clients and if you have another set of data stored in an Excel or CSV file, then you can select that dataset from the **Override** list.



Remember: You must have uploaded the dataset as an Excel or CSV file into the Git repository, and ensured that both the original dataset (from the test asset) and new datasets (added to the project) have the same column names.

c. Select the dataset that you want to use in the test run from any of the following options:

- Select the dataset that is displayed as the default dataset when the test asset contains a single dataset.



Note: If there is only one dataset in the test asset, then that dataset is displayed as the default dataset.

- Select the dataset from the list.



Note: If there are multiple datasets in the test asset, the datasets are listed in their increasing alphabetical order.

- Select the dataset from the **Override** list to override the dataset that was associated with the test in the desktop client.





Important: If the test contains an encrypted dataset, the Project Owner must classify it in the **DATA SECURITY** tab on the Project page before you can select it. You must have added datasets to your project from the **Dataset** page for the datasets to be displayed in the **Override** list.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

12. Follow the instructions in this step if the test requires a variable that must be passed to the test at the test run time.

You must configure the supported browser by using a variable if the test has a browser configured, which is different from the one that is supported by Rational® Test Automation Server.

- Click the **VARIABLES** tab, if it is not already open.
- Choose one of the following methods to add the variables:

- To add new variables manually, click the **Add Variable** icon , enter the name, and value of the variable.
 - i. Enter `RTW_WebUI_Browser_Selection` as the name.
 - ii. Enter `Firefox` as the value, if you want to use Firefox as the browser and override the browser that is specified in the HTML test.
- To add new variables from your local computer or from the Git repository that is associated with your server project, click the **Upload** icon  and select the **Upload from local system** or **Browse from server** to select the variable file.



Note: You must have created a file with the variables before you can select the file.


The default value for the variables is null or an empty field.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

13. Follow the instructions if you want to export the test results to a Jira issue in your Xray project in Jira:

- a. Click the **RESULTS** tab.
- b. Perform the actions listed in the following table to select an option to export the test results:


Export options	Results in...	Actions
Jira Xray	The test results of the test that you configure for a run are exported to the selected issue after the test run is completed. The results are exported to the selected issue that exists in your Xray project in Jira.	Perform the following steps: <ol style="list-style-type: none"> i. Select the Jira Xray option. <p>The Select issue field is displayed.</p> <ol style="list-style-type: none"> ii. Click the field to view the list of issues in the Xray project in Jira. Alternatively, you can enter the issue key in the Jira Xray project. iii. Select the issue to which you want to export the test results. iv. Continue with the next step.
New Jira item	The test results of the test that you configure for a run are exported to a new Jira issue. The issue is created in your project in Jira after the test run is completed.	Perform the following steps: <ol style="list-style-type: none"> i. Select the New Jira item option. ii. Continue with the next step.

Export options	Results in...	Actions
None	<p>The test results are not exported.</p> <p> Note: This is the default option.</p>	<p>Perform the following steps when:</p> <ul style="list-style-type: none"> ▪ You do not want to export the test results. ▪ You selected a Jira issue in a previous run of the same test, and you do not want to export results of the current run. <ul style="list-style-type: none"> i. Select the None option. ii. Continue with the next step.

14. Follow the instructions if you want to change the location of a Kubernetes cluster for running the test:

a. Click the **LOCATION** tab, if it is not already open.

The **Default Cluster** is the default location where the test runs, and it is listed under the **Host** column. The information about the availability of the default location is displayed.

 **Important:** You must have added the following remote hosts to your project that are then displayed under the **Override** column:

- Docker hosts that are registered with Rational® Test Automation Server.
- Kubernetes clusters that are registered with a team space.

 **Notes:**

- If the remote Docker hosts or clusters are not added to your project, the option **No override options** is displayed as the default value and the test runs in the Kubernetes cluster of Rational® Test Automation Server.
- If remote Docker hosts or clusters are added to your project, the added Docker hosts or clusters are displayed along with their availability status and ownership information.

b. Select the location where you want to run the test asset from the following options:

- Select the **Default Cluster** when no remote Docker hosts or clusters are available in your project.
- Select the remote Docker host or cluster from the list when a remote Docker host or cluster is available in your project.
- Select **No override options**, if you selected any remote Docker host or cluster and want to revert to the **Default Cluster** to run the test asset.

c. Click **Execute**.


The test run is initiated.

Results

You have started or scheduled a test run of a Compound Test that contains Functional HTML tests.

What to do next

You can choose to perform any of the following tasks:

- Stop the test run at any point after the test run is initiated from the **Execution** page. See [Stopping test runs on page 403](#).
- Cancel a scheduled test run from the **Execution** page. See [Canceling scheduled test runs on page 406](#).
- View all the states of the test asset by clicking the **Show in the Progress page** icon  for the test asset for which you started or scheduled the run. See [Viewing the state of test assets on page 394](#).
- View the progress of the test from the **Progress** page. See [Viewing the progress of running test assets on page 398](#).
- View the results, reports, and logs of the test from the **Results** page after the test completes the run. See [Test results on page 455](#).

Related information

[Resetting the configuration settings for a test run on page 402](#)

[Test run configurations on page 249](#)

Configuring a run of a Compound Test that contains Web UI tests or a Functional Web UI test

After you added the test resources that you created in the desktop client to the project, you can configure a Compound Test that contains Web UI tests or a Functional Web UI test to be run on IBM® Rational® Test Automation Server or a remote Kubernetes cluster.

Before you begin

You must have completed the following tasks:

- Read and completed the tasks mentioned in [Prerequisites to running tests on page 198](#), if they apply to the test that you want to configure for a run.
- Read and completed the tasks mentioned in [Test run considerations for running tests on remote performance agents on page 211](#), if you want to use a remote agent as the location to run the test.
- Read and completed the tasks mentioned in [Considerations for using a remote Kubernetes cluster on page 235](#), if you want to use a remote Kubernetes cluster as a location to run the test.
- Completed the Xray integration, if you want to export the test results to a Jira issue in your Xray project in Jira. See [Integration with Xray test management for Jira on page 593](#).
- Created a Compound test that contains Web UI tests or a Functional Web UI test in Rational® Functional Tester and added the test asset to the project repository on IBM® Rational® Test Automation Server.
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).

- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created a project in your team space in Rational® Test Automation Server. See [Adding a project on page 619](#).
- Added the remote Git repository that contains the test resources to the project. See [Adding repositories to a server project on page 620](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the test assets, and then click **Execution**.
3. Select the branch of the repository that contains the test assets.

The test assets that are contained in the selected branch of the repository are displayed in the following tabs on the **Execution** page:

Tab	Description
SUITES	Lists all suites, compound tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules that are in the selected branch of the repository.
TESTS	Lists all API tests, functional tests, or performance tests that are in the selected branch of the repository.
ADVANCED	Lists all assets that are displayed when custom filters are applied for assets in the selected branch of the repository.

4. Select the tab based on the type of test assets that you want to run as indicated in the following table:


If...	Then...
You want to run AFT Suites, API Suites, Compound Tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules,.	Click the SUITES tab.
You want to run API tests, functional tests, or performance tests.	Click the TESTS tab.
You want to find Suites and tests by using custom filters, and then run Suites and tests.	Click the ADVANCED tab.





































The test assets in the selected tab are displayed.


5. Identify the test asset that you want to run by performing any of the following actions:

- Scroll through the list.


 **Tip:** You can hover over the icon in the **Type** column to know the type of the test asset.

 **Note:** You can also identify the type of the asset from the icon that represents the test type as shown in the following table:

Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the AD-VANCED tab
	API test			
	Functional test			
	Performance test			
	AFT Suite			
	API Suite			
	Compound Test			
	HCL AppScan CodeSweep			
	JMeter Test			
	JUnit Test			
	Postman test			
	Rate Schedule			
	VU Schedule			

- Search for the test asset by entering any text contained in the test asset name in the **Search** field.
- Click the **Filter** icon  in the **SUITES** or **TESTS** tab to filter the displayed assets based on the asset type.

For example, select **API Suite** in the **SUITES** tab to display all API Suites or select **Functional Test** in the **TESTS** tab to display all functional tests that are in the selected branch of the repository.

- Click the **Filter** icon  in the **ADVANCED** tab, and then create a filter query by using the **New filter** option by performing the following steps:
 - a. Click **New filter**.
 - b. Enter a name for the filter.
 - c. Select an operator, and add a rule or a group of rules.
 - d. Add or enter the relevant parameters and either select or enter the condition and the criteria for the condition.

You can select a parameter from the following list:

- Type
 - Test Asset Name
 - Test Asset Path
 - Last Result
 - Next Run
 - Components
- e. Save the filter query to save and apply the filter query to filter the assets based on the query.

The test assets that match the filter criteria are displayed.

- Retrieve and apply a saved filter query, if you have saved filter queries previously by performing the following steps:



Note: The filter query applied previously is selected and the assets based on that filter query are displayed. To apply a different filter query, you must have created and saved the filter query.

- a. Click the **Filter** icon  in the **ADVANCED** tab.


The filter queries that you created and saved are displayed.

- b. Click the filter that you want to apply.

The test assets that match the filter criteria are displayed.

You have identified the test asset that you want to run.

Clicking the test name displays the **Details** panel. You can view the details of the test such as the description, the branch in the repository that contains the asset, the Git details, and the details of the commits to the repository. You can also view the history of the test runs for the specific test under the **History** tab of the **Details** panel.

6. Click the **Execute** icon  in the row of the identified test asset.

The **Execute test asset** dialog is displayed.



Notes:

- If you have configured some or all of the settings for the current test run, and you do not want to continue with those settings, you can reset the settings by clicking **Reset**.
- If you want to repeat a test run and do not want to use the saved settings from a previous run, you can reset all the saved settings to their default values by clicking **Reset**.

7. Select the version of the test resources that you want to run by completing any of the following actions:



Note: The test resources in the version can contain the test assets, datasets, AFT XML files, API environment tags, and other resources specific to projects created from any of the desktop clients.

- Expand the list in the **Version** field, find the version of the test resources, and then select the version.

Use the following details about the version of the test resources that are displayed to identify the version that you want:

- Commit message.
- Tags labeled by the user for the version committed.
- The user who committed the version to the repository.
- Relative time of the commit. For example, *2 hours ago* or *3 days ago*.

The list displays the versions of the test resources committed by all users to the branch in the repository. The versions are arranged with the latest version that is committed, and then followed by the versions committed previously.

- Expand the list in the **Version** field, and then search for the version that you want to select by entering a partial or the complete commit message of that version.

The version that matches the search criteria is displayed and it is selected for the test run.

The default value for the version selected for the run is the latest version in the selected branch of the repository. If you do not select any version, then the latest version is selected for the test run.



Notes:



- If you selected a version but you do not want to use that version in the test run, you can remove the selected version by clicking the **✕** icon. As a result, the default version is selected for the test run.
- If you repeated a test or ran the test again from the **Results** page, then the version of the test resources that you chose for the earlier run is shown as selected. You can either retain this version or select any other version from the list. You can also remove the previous version by clicking the **✕** icon.

8. Select the time for scheduling the test run from the following options:

- No action is required if you want to initiate the test run immediately after you click **Execute**.






Important: Click **Execute** only after you have configured the other settings in this dialog.



Note: The default time for scheduling a run is **Now**.

- Select **Schedule** and perform the following actions if you want to schedule a single test run or configure recurring test runs:


Schedule	Actions
A single test run	<p>Perform the following steps:</p> <ol style="list-style-type: none"> a. Click the Calendar icon  in the row of the Start field. b. Select the date. c. Select the time at which the test run must start. d. Select the Never option in the Repeat every field. e. Click Save. <p>The test run is scheduled to run at the selected time on the scheduled date.</p>
Recurring test runs	<p>Perform the following steps:</p> <ol style="list-style-type: none"> a. Click the Calendar icon  in the row of the Start field. b. Select the date. c. Select the time at which the test run must start. d. Set the frequency at which the test runs must run by entering the number in the Repeat every field, and then select the period from the list. You can select from the following options: <ul style="list-style-type: none"> ▪ Minute ▪ Hour ▪ Day

Schedule	Actions
	<ul style="list-style-type: none"> ▪ Week ▪ Month <p>For example, if you want the test run to be run every day at the set time, enter 1 and, then select the Day(s) option.</p> <p>You can also schedule the test to run at the set time on specific weekdays by selecting the Week(s) option. For example, the following image displays the days selected for the test runs as <i>Monday, Wednesday, and Friday</i>:</p> <div data-bbox="708 632 1421 793" style="border: 1px solid black; padding: 5px; margin: 10px 0;"> <p>Repeat every <input type="text" value="1"/> <input type="text" value="Week(s)"/> ▾</p> <p style="text-align: center;"> <input type="radio"/> S <input checked="" type="radio"/> M <input type="radio"/> T <input checked="" type="radio"/> W <input type="radio"/> T <input checked="" type="radio"/> F <input type="radio"/> S </p> </div> <ul style="list-style-type: none"> e. Set the option to end the recurring test runs from the following options: <ul style="list-style-type: none"> ▪ Select the Never option, if you do not want the recurring test runs to end. ▪ Select the On option, and then click the Calendar icon . You can select the date and time after which the scheduled test runs do not run. f. Click Save. <p>The recurring test runs are scheduled to start the first run at the selected time on the scheduled date, and to repeat the run at the frequency that you set.</p>

9. Enter a label for the test run that helps you to identify the test on the **Results** page.

After the test run completes, the label that you entered is displayed for the test under the **Labels** column on the **Results** page. After you have created a label, any member of the project can use that label.

The default value for the **Label** field is null or an empty field.

 **Important:** The configuration that you set for the test run in the **Execute test asset** dialog is preserved when you run the same test again. Those changes are not visible when another user logs in to Rational® Test Automation Server. For example, if you created new variables on the server, those variables are available only for you when the same test is run again.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

10. Click **Advanced** to make the following advanced configurations:

- a. Enter any JVM arguments that must be passed to the test run at run time in the **JVM Arguments** field, if applicable for the test.

For example, you can set a maximum Java heap size.

- b. Enter program arguments that must be passed to the test run at run time in the **Program Arguments**, if applicable for the test.

If the test that you want to configure supports the presentation of the test results data in different formats, then do the required tasks in the following scenarios:

If...	Then...
You want the test results data in different supported formats.	Enter <code>-history cisterna</code> in the Program Arguments field.
You want to obtain the test results as a Jaeger trace.	Enter <code>-history jaeger</code> in the Program Arguments field.
You want the report as a test log and as a Jaeger trace.	Enter <code>-history jaeger, testlog</code> in the Program Arguments field.
Jaeger is already set as a program argument and you want to remove the Jaeger trace for your test.	Delete the <code>-history jaeger</code> entry in the Program Arguments field.



Note: The default report format is the *test log* format for the test reports.

- c. Enter the environment variables that must be passed to the test run at run time in the **Environment Variables** field, if applicable for the test.

For example, enter the environment variables when the third-party libraries that are used in the test run refer to the environment variables for configuration.



Note: You must separate the arguments or variables with a white space when you enter them in the same line or start each argument or variable on a new line.

The default value for each of the fields for the advanced settings is null or an empty field.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

11. Follow the instructions if you are running a test asset that contains datasets:

a. Click the **DATA SOURCES** tab, if it is not already open.

b. Consider the following information about datasets before you select a dataset:

The default value for the datasets in the **DATA SOURCES** tab is null if the test asset did not have an associated dataset. If the asset had an associated dataset, the default value is the associated dataset.

You can utilize the dataset stored as an Excel or CSV file to override the original dataset associated with the Suite, test, or schedule. For example, when you have associated a dataset in `.xlsx`, `.xls`, or `.csv` format with the test or schedule in desktop clients and if you have another set of data stored in an Excel or CSV file, then you can select that dataset from the **Override** list.



Remember: You must have uploaded the dataset as an Excel or CSV file into the Git repository, and ensured that both the original dataset (from the test asset) and new datasets (added to the project) have the same column names.

c. Select the dataset that you want to use in the test run from any of the following options:

- Select the dataset that is displayed as the default dataset when the test asset contains a single dataset.



Note: If there is only one dataset in the test asset, then that dataset is displayed as the default dataset.

- Select the dataset from the list.



Note: If there are multiple datasets in the test asset, the datasets are listed in their increasing alphabetical order.

- Select the dataset from the **Override** list to override the dataset that was associated with the test in the desktop client.





Important: If the test contains an encrypted dataset, the Project Owner must classify it in the **DATA SECURITY** tab on the Project page before you can select it. You must have added datasets to your project from the **Dataset** page for the datasets to be displayed in the **Override** list.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

12. Follow the instructions in this step if the test requires a variable that must be passed to the test at the test run time.

You must configure the supported browser by using a variable if the test has a browser configured, which is different from the one that is supported by Rational® Test Automation Server.

- a. Click the **VARIABLES** tab, if it is not already open.
- b. Choose one of the following methods to add the variables:
 - To add new variables manually, click the **Add Variable** icon , enter the name, and value of the variable.
 - i. Enter `RTW_WebUI_Browser_Selection` as the name.
 - ii. Enter `Firefox` as the value, if you want to use Firefox as the browser and override the browser that is specified in the Web UI test.
 - To add new variables from your local computer or from the Git repository that is associated with your server project, click the **Upload** icon  and select the **Upload from local system** or **Browse from server** to select the variable file.



Note: You must have created a file with the variables before you can select the file.

The default value for the variables is null or an empty field.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

13. Select any of the following steps that depends on the location to run the test:
 - If you want to run the tests on the remote agents, go to [Step 14 on page 300](#).
 - If you want to run the tests on a Kubernetes cluster, go to [Step 16 on page 302](#).
14. Follow the instructions if you want to run a compound test on the remote performance agent that is connected either to the default Kubernetes cluster or the remote Kubernetes cluster:
 - a. Click the **LOCATION** tab, if it is not already open.

The performance agents that are configured in the test asset are listed under the **Host** column. The information about the availability and capabilities of the agent are displayed.



Note: You must have added performance agents to your project from the **Agents and Intercepts** page for the performance agents to be displayed under the **Override** column.

The default value for the performance agents is null or an empty field if no performance agents were configured in the test asset. If the test asset contains performance agents that are configured, then the default performance agent is the first item to be displayed on the list of performance agents listed in the increasing alphabetical order.

- b. Select the performance agent where you want to run the test asset in the following scenarios:


If...	Then...	Action
The performance agent that is specified in the test assets is available and no other performance agents are added to the project.	The performance agent that is specified in the test assets is displayed in the Override column.	Select the performance agent in the Host that is also displayed in the Override column as the location to run the test.
The performance agent that is specified in the test assets is not available and no other performance agents are added to the project.	There is no performance agent displayed for override in the Override column.	You cannot run the test on the performance agent. You must wait until the performance agent is available to run the test on the performance agent.
The performance agent that is specified in the test assets is available and there are other performance agents that are added to the project.	Performance agents that have capabilities that match with the performance agent capabilities specified in the test assets are listed in the Override column as follows: <ul style="list-style-type: none"> ▪ Performance agents with capabilities that best match the capabilities in the agent in the asset are at the top of the list. ▪ Performance agents with capabilities that do not match with the capabilities in the agent in the asset are listed subsequently. You can find details of both performance agents for the matching capabilities when you hover over the agent in the Override column.	Perform any of the following actions: <ul style="list-style-type: none"> ▪ Select the performance agent in the Host that is also displayed in the Override column as the location to run the test. ▪ Select a performance agent in the Override column as the location to run the test.

c. Click **Execute**.

15. Follow the instructions if you want to export the test results to a Jira issue in your Xray project in Jira:

a. Click the **RESULTS** tab.


b. Perform the actions listed in the following table to select an option to export the test results:

Export options	Results in...	Actions
Jira Xray	<p>The test results of the test that you configure for a run are exported to the selected issue after the test run is completed. The results are exported to the selected issue that exists in your Xray project in Jira.</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> i. Select the Jira Xray option. <p>The Select issue field is displayed.</p> <ol style="list-style-type: none"> ii. Click the field to view the list of issues in the Xray project in Jira. Alternatively, you can enter the issue key in the Jira Xray project. iii. Select the issue to which you want to export the test results. iv. Continue with the next step.
New Jira item	<p>The test results of the test that you configure for a run are exported to a new Jira issue. The issue is created in your project in Jira after the test run is completed.</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> i. Select the New Jira item option. ii. Continue with the next step.
None	<p>The test results are not exported.</p> <p> Note: This is the default option.</p>	<p>Perform the following steps when:</p> <ul style="list-style-type: none"> ▪ You do not want to export the test results. ▪ You selected a Jira issue in a previous run of the same test, and you do not want to export results of the current run. <ol style="list-style-type: none"> i. Select the None option. ii. Continue with the next step.

16. Follow the instructions if you want to change the location of a Kubernetes cluster for running the test:

a. Click the **LOCATION** tab, if it is not already open.

The **Default Cluster** is the default location where the test runs, and it is listed under the **Host** column. The information about the availability of the default location is displayed.

 **Important:** You must have added the following remote hosts to your project that are then displayed under the **Override** column:



- Docker hosts that are registered with Rational® Test Automation Server.
- Kubernetes clusters that are registered with a team space.

**Notes:**

- If the remote Docker hosts or clusters are not added to your project, the option **No override options** is displayed as the default value and the test runs in the Kubernetes cluster of Rational® Test Automation Server.
- If remote Docker hosts or clusters are added to your project, the added Docker hosts or clusters are displayed along with their availability status and ownership information.

b. Select the location where you want to run the test asset from the following options:

- Select the **Default Cluster** when no remote Docker hosts or clusters are available in your project.
- Select the remote Docker host or cluster from the list when a remote Docker host or cluster is available in your project.
- Select **No override options**, if you selected any remote Docker host or cluster and want to revert to the **Default Cluster** to run the test asset.

c. Click **Execute**.


The test run is initiated.

Results

You have started or scheduled a test run of a Compound Test that contains Web UI tests or a Functional Web UI test.

What to do next

You can choose to perform any of the following tasks:

- Stop the test run at any point after the test run is initiated from the **Execution** page. See [Stopping test runs on page 403](#).
- Cancel a scheduled test run from the **Execution** page. See [Canceling scheduled test runs on page 406](#).
- View all the states of the test asset by clicking the **Show in the Progress page** icon  for the test asset for which you started or scheduled the run. See [Viewing the state of test assets on page 394](#).
- View the progress of the test from the **Progress** page. See [Viewing the progress of running test assets on page 398](#).
- View the results, reports, and logs of the test from the **Results** page after the test completes the run. See [Test results on page 455](#).

Related information

[Resetting the configuration settings for a test run on page 402](#)

[Test run configurations on page 249](#)

Configuring a run of a Compound Test that contains mobile tests or a Functional Mobile test

After you added the test resources that you created in the desktop client to the project, you can configure a Compound Test that contains mobile tests or a Functional mobile test to be run on IBM® Rational® Test Automation Server or a remote Kubernetes cluster.

Before you begin

You must have completed the following tasks:

- Created a Compound Test or a Functional test that contains the mobile tests in Rational® Functional Tester and added the test asset to the project repository on IBM® Rational® Test Automation Server.
- Created a variable file if you want to import the variables file. The file must contain the details of the UI Test agent or the device cloud to which the mobile device is connected.
- Read [Considerations for using Jaeger traces in reports on page 205](#), if you want to configure a run and you want Jaeger to report the test results for the test.
- Read and completed the tasks mentioned in [Test run considerations for running tests on remote performance agents on page 211](#), if you want to use a remote agent as the location to run the test.
- Read and completed the tasks mentioned in [Considerations for using a remote Kubernetes cluster on page 235](#), if you want to use a remote Kubernetes cluster as a location to run the test.
- Completed the Xray integration, if you want to export the test results to a Jira issue in your Xray project in Jira. See [Integration with Xray test management for Jira on page 593](#).
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created a project in your team space in Rational® Test Automation Server. See [Adding a project on page 619](#).
- Added the remote Git repository that contains the test resources to the project. See [Adding repositories to a server project on page 620](#).

About this task

You can run the mobile tests that are contained in a Compound Test or a Functional test after you create the tests in Rational® Functional Tester, and then add them to your project on IBM® Rational® Test Automation Server. You can then run the mobile tests on the mobile devices that are connected to any of the following agents or clouds:

- Remote UI Test agents. Requires the variables at test run time as specified in [Table 6: Variables for the UI Test agent on page 305](#).
- The BitBar cloud. Requires the variables at test run time as specified in [Table 7: Variables for the BitBar cloud on page 305](#).
- The Perfecto mobile cloud. Requires the variables at test run time as specified in [Table 8: Variables for the Perfecto mobile cloud on page 306](#).
- The pCloudy mobile cloud. Requires the variables at test run time as specified in [Table 9: Variables for the pCloudy mobile cloud on page 306](#).

You must provide the details of the UI Test agent or the mobile cloud to which the mobile devices are connected as variables in [Step 12 on page 314](#). You can either enter the variables or use the file in which you entered the variables. You must refer to the following tables for the variables that are required for a successful run:

Table 6. Variables for the UI Test agent

Name of the Variable	Action for the Value field
Mobile_Device_Selection	Specify the name of the mobile device that is connected to the UI Test agent.
appium.server.host	Specify the host name or IP address of the UI Test agent.
appium.server.port	Specify the port on the UI Test agent that is configured to communicate with Rational® Test Automation Server.

Table 7. Variables for the BitBar cloud

Name of the Variable	Value
Mobile_Device_Selection	Specify the name of the mobile device that is connected to the BitBar cloud.
bitbar.apikey	Specify the user token generated for your BitBar account to authenticate your connection with the BitBar cloud.
bitbar.host	Specify the host name of the BitBar cloud instance.
bitbar.project	Specify the name of the project that contains the recorded test.

Table 7. Variables for the BitBar cloud (continued)

Name of the Variable	Value
bitbar.testrun	Specify a name for the test run that must be displayed in the BitBar dashboard for the test run.

Table 8. Variables for the Perfecto mobile cloud

Name of the Variable	Value
Mobile_Device_Selection	Specify the name of the mobile device that is connected to the Perfecto mobile cloud.
perfecto.securitytoken	Specify the user token generated for your Perfecto account to authenticate your connection with the Perfecto mobile cloud.
perfecto.host	Specify the host name of the Perfecto mobile cloud instance.

Table 9. Variables for the pCloudy mobile cloud

Name of the Variable	Value
Mobile_Device_Selection	Specify the name of the mobile device that is connected to the pCloudy mobile cloud.
pcloudy.apikey	Specify the API key that is generated for your pCloudy account to authenticate your connection with the pCloudy mobile cloud.
pcloudy.username	Specify your username with which you can access the pCloudy mobile cloud instance.
pcloudy.host	Specify the host name of the pCloudy mobile cloud instance.

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the test assets, and then click **Execution**.
3. Select the branch of the repository that contains the test assets.

The test assets that are contained in the selected branch of the repository are displayed in the following tabs on the **Execution** page:

Tab	Description
SUITES	Lists all suites, compound tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules that are in the selected branch of the repository.
TESTS	Lists all API tests, functional tests, or performance tests that are in the selected branch of the repository.
ADVANCED	Lists all assets that are displayed when custom filters are applied for assets in the selected branch of the repository.

4. Select the tab based on the type of test assets that you want to run as indicated in the following table:


If...	Then...
You want to run AFT Suites, API Suites, Compound Tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules,.	Click the SUITES tab.
You want to run API tests, functional tests, or performance tests.	Click the TESTS tab.
You want to find Suites and tests by using custom filters, and then run Suites and tests.	Click the ADVANCED tab.


The test assets in the selected tab are displayed.





































5. Identify the test asset that you want to run by performing any of the following actions:


- Scroll through the list.

 **Tip:** You can hover over the icon in the **Type** column to know the type of the test asset.


 **Note:** You can also identify the type of the asset from the icon that represents the test type as shown in the following table:



Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the ADVANCED tab
	API test			
	Functional test			
	Performance test			
	AFT Suite			
	API Suite			
	Compound Test			
	HCL AppScan CodeSweep			
	JMeter Test			
	JUnit Test			
	Postman test			
	Rate Schedule			
	VU Schedule			

- Search for the test asset by entering any text contained in the test asset name in the **Search** field.
- Click the **Filter** icon  in the **SUITES** or **TESTS** tab to filter the displayed assets based on the asset type.

For example, select **API Suite** in the **SUITES** tab to display all API Suites or select **Functional Test** in the **TESTS** tab to display all functional tests that are in the selected branch of the repository.

- Click the **Filter** icon  in the **ADVANCED** tab, and then create a filter query by using the **New filter** option by performing the following steps:
 - Click **New filter**.
 - Enter a name for the filter.

- c. Select an operator, and add a rule or a group of rules.
- d. Add or enter the relevant parameters and either select or enter the condition and the criteria for the condition.

You can select a parameter from the following list:

- Type
 - Test Asset Name
 - Test Asset Path
 - Last Result
 - Next Run
 - Components
- e. Save the filter query to save and apply the filter query to filter the assets based on the query.

The test assets that match the filter criteria are displayed.

- Retrieve and apply a saved filter query, if you have saved filter queries previously by performing the following steps:



Note: The filter query applied previously is selected and the assets based on that filter query are displayed. To apply a different filter query, you must have created and saved the filter query.

- a. Click the **Filter** icon  in the **ADVANCED** tab.


The filter queries that you created and saved are displayed.

- b. Click the filter that you want to apply.

The test assets that match the filter criteria are displayed.

You have identified the test asset that you want to run.

Clicking the test name displays the **Details** panel. You can view the details of the test such as the description, the branch in the repository that contains the asset, the Git details, and the details of the commits to the repository. You can also view the history of the test runs for the specific test under the **History** tab of the **Details** panel.

6. Click the **Execute** icon  in the row of the identified test asset.

The **Execute test asset** dialog is displayed.



Notes:



- If you have configured some or all of the settings for the current test run, and you do not want to continue with those settings, you can reset the settings by clicking **Reset**.
- If you want to repeat a test run and do not want to use the saved settings from a previous run, you can reset all the saved settings to their default values by clicking **Reset**.

7. Select the version of the test resources that you want to run by completing any of the following actions:



Note: The test resources in the version can contain the test assets, datasets, AFT XML files, API environment tags, and other resources specific to projects created from any of the desktop clients.

- Expand the list in the **Version** field, find the version of the test resources, and then select the version.

Use the following details about the version of the test resources that are displayed to identify the version that you want:

- Commit message.
- Tags labeled by the user for the version committed.
- The user who committed the version to the repository.
- Relative time of the commit. For example, *2 hours ago* or *3 days ago*.

The list displays the versions of the test resources committed by all users to the branch in the repository. The versions are arranged with the latest version that is committed, and then followed by the versions committed previously.

- Expand the list in the **Version** field, and then search for the version that you want to select by entering a partial or the complete commit message of that version.

The version that matches the search criteria is displayed and it is selected for the test run.

The default value for the version selected for the run is the latest version in the selected branch of the repository. If you do not select any version, then the latest version is selected for the test run.



Notes:

- If you selected a version but you do not want to use that version in the test run, you can remove the selected version by clicking the ✕ icon. As a result, the default version is selected for the test run.
- If you repeated a test or ran the test again from the **Results** page, then the version of the test resources that you chose for the earlier run is shown as selected. You can either retain this version or select any other version from the list. You can also remove the previous version by clicking the ✕ icon.

8. Select the time for scheduling the test run from the following options:

- No action is required if you want to initiate the test run immediately after you click **Execute**.







Important: Click **Execute** only after you have configured the other settings in this dialog.



Note: The default time for scheduling a run is **Now**.

- Select **Schedule** and perform the following actions if you want to schedule a single test run or configure recurring test runs:


Schedule	Actions
A single test run	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Select the Never option in the Repeat every field. Click Save. <p>The test run is scheduled to run at the selected time on the scheduled date.</p>
Recurring test runs	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Set the frequency at which the test runs must run by entering the number in the Repeat every field, and then select the period from the list. You can select from the following options: <ul style="list-style-type: none"> ▪ Minute ▪ Hour ▪ Day ▪ Week ▪ Month <p>For example, if you want the test run to be run every day at the set time, enter 1 and, then select the Day(s) option.</p> <p>You can also schedule the test to run at the set time on specific weekdays by selecting the Week(s) option. For example, the following image displays the days selected for the test runs as <i>Monday, Wednesday, and Friday</i>:</p>

Schedule	Actions
	<div data-bbox="708 262 1419 422" style="border: 1px solid black; padding: 5px; margin-bottom: 10px;"> Repeat every <input type="text" value="1"/> <input type="text" value="Week(s)"/>  <div style="display: flex; justify-content: center; gap: 10px;"> S M T W T F S </div> </div> <p>e. Set the option to end the recurring test runs from the following options:</p> <ul style="list-style-type: none"> ▪ Select the Never option, if you do not want the recurring test runs to end. ▪ Select the On option, and then click the Calendar icon . You can select the date and time after which the scheduled test runs do not run. <p>f. Click Save.</p> <p>The recurring test runs are scheduled to start the first run at the selected time on the scheduled date, and to repeat the run at the frequency that you set.</p>

9. Enter a label for the test run that helps you to identify the test on the **Results** page.

After the test run completes, the label that you entered is displayed for the test under the **Labels** column on the **Results** page. After you have created a label, any member of the project can use that label.

The default value for the **Label** field is null or an empty field.

 **Important:** The configuration that you set for the test run in the **Execute test asset** dialog is preserved when you run the same test again. Those changes are not visible when another user logs in to Rational® Test Automation Server. For example, if you created new variables on the server, those variables are available only for you when the same test is run again.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

10. Click **Advanced** to make the following advanced configurations:

a. Enter any JVM arguments that must be passed to the test run at run time in the **JVM Arguments** field, if applicable for the test.

For example, you can set a maximum Java heap size.

b. Enter program arguments that must be passed to the test run at run time in the **Program Arguments**, if applicable for the test.

If the test that you want to configure supports the presentation of the test results data in different formats, then do the required tasks in the following scenarios:

If...	Then...
You want the test results data in different supported formats.	Enter <code>-history cisterna</code> in the Program Arguments field.
You want to obtain the test results as a Jaeger trace.	Enter <code>-history jaeger</code> in the Program Arguments field.
You want the report as a test log and as a Jaeger trace.	Enter <code>-history jaeger, testlog</code> in the Program Arguments field.
Jaeger is already set as a program argument and you want to remove the Jaeger trace for your test.	Delete the <code>-history jaeger</code> entry in the Program Arguments field.



Note: The default report format is the *test log* format for the test reports.

- c. Enter the environment variables that must be passed to the test run at run time in the **Environment Variables** field, if applicable for the test.

For example, enter the environment variables when the third-party libraries that are used in the test run refer to the environment variables for configuration.



Note: You must separate the arguments or variables with a white space when you enter them in the same line or start each argument or variable on a new line.

The default value for each of the fields for the advanced settings is null or an empty field.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

11. Follow the instructions if you are running a test asset that contains datasets:

- a. Click the **DATA SOURCES** tab, if it is not already open.
- b. Consider the following information about datasets before you select a dataset:

The default value for the datasets in the **DATA SOURCES** tab is null if the test asset did not have an associated dataset. If the asset had an associated dataset, the default value is the associated dataset.

You can utilize the dataset stored as an Excel or CSV file to override the original dataset associated with the Suite, test, or schedule. For example, when you have associated a dataset in `.xlsx`, `.xls`, or `.csv` format with the test or schedule in desktop clients and if you have another set of data stored in an Excel or CSV file, then you can select that dataset from the **Override** list.



Remember: You must have uploaded the dataset as an Excel or CSV file into the Git repository, and ensured that both the original dataset (from the test asset) and new datasets (added to the project) have the same column names.

c. Select the dataset that you want to use in the test run from any of the following options:

- Select the dataset that is displayed as the default dataset when the test asset contains a single dataset.



Note: If there is only one dataset in the test asset, then that dataset is displayed as the default dataset.

- Select the dataset from the list.



Note: If there are multiple datasets in the test asset, the datasets are listed in their increasing alphabetical order.

- Select the dataset from the **Override** list to override the dataset that was associated with the test in the desktop client.





Important: If the test contains an encrypted dataset, the Project Owner must classify it in the **DATA SECURITY** tab on the Project page before you can select it. You must have added datasets to your project from the **Dataset** page for the datasets to be displayed in the **Override** list.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

12. Perform the following steps to provide the variables that specify the UI Test agent or cloud to which the mobile device is attached. You can either enter the variables that must be passed to the test at the test run time or import the file that contains the variables.

- a. Click the **VARIABLES** tab, if it is not already open.
- b. Choose one of the following methods to add the variables:

- To add new variables manually, click the **Add Variable** icon , enter the name, and value of the variable.
- To add new variables from your local computer or from the Git repository that is associated with your server project, click the **Upload** icon  and select the **Upload from local system** or **Browse from server** to select the variable file.



Note: You must have created a file with the variables before you can select the file.

13. Select any of the following steps that depends on the location to run the test:
 - If you want to run the tests on the remote agents, go to [Step 14 on page 315](#).
 - If you want to run the tests on a Kubernetes cluster, go to [Step 16 on page 317](#).
14. Follow the instructions if you want to run a test on the remote performance agent that is connected either to the default Kubernetes cluster or the remote Kubernetes cluster:
 - a. Click the **LOCATION** tab, if it is not already open.

The performance agents that are configured in the test asset are listed under the **Host** column. The information about the availability of the performance agent is displayed.



Note: You must have added performance agents to your project from the **Agents and Intercepts** page for the performance agents to be displayed under the **Override** column.

The default value for the performance agents is null or an empty field if no performance agents were configured in the test asset. If the test asset contains performance agents that are configured, then the default performance agent is the first item to be displayed on the list of performance agents listed in the increasing alphabetical order.

- b. Select the performance agent where you want to run the test asset in the following scenarios:

If...	Then...	Action
The performance agent that is specified in the test assets is available and no other performance agents are added to the project.	The performance agent that is specified in the test assets is displayed in the Override column.	Select the performance agent in the Host that is also displayed in the Override column as the location to run the test.
The performance agent that is specified in the test assets is not available and no other performance agents are added to the project.	There is no performance agent displayed for override in the Override column.	You cannot run the test on the performance agent. You must wait until the performance agent is available to run the test on the agent.

If...	Then...	Action
<p>The agent that is specified in the test assets is available and there are other agents that are added to the project.</p>	<p>Performance agents that have capabilities that match with the agent capabilities specified in the test assets are listed in the Override column as follows:</p> <ul style="list-style-type: none"> ▪ Performance agents with capabilities that best match the capabilities in the agent in the asset are at the top of the list. ▪ Performance agents with capabilities that do not match with the capabilities in the agent in the asset are listed subsequently. <p>You can find details of both performance agents for the matching capabilities when you hover over the performance agent in the Override column.</p>	<p>Perform any of the following actions:</p> <ul style="list-style-type: none"> ▪ Select the performance agent in the Host that is also displayed in the Override column as the location to run the test. ▪ Select a performance agent in the Override column as the location to run the test.

c. Click **Execute**.


The test run is initiated.

15. Follow the instructions if you want to export the test results to a Jira issue in your Xray project in Jira:

a. Click the **RESULTS** tab.

b. Perform the actions listed in the following table to select an option to export the test results:

Export options	Results in...	Actions
<p>Jira Xray</p>	<p>The test results of the test that you configure for a run are exported to the selected issue after the test run is completed. The results are exported to the selected issue that exists in your Xray project in Jira.</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> i. Select the Jira Xray option. <p>The Select issue field is displayed.</p> <ol style="list-style-type: none"> ii. Click the field to view the list of issues in the Xray project in Jira. Alternatively, you

Export options	Results in...	Actions
		<p>can enter the issue key in the Jira Xray project.</p> <p>iii. Select the issue to which you want to export the test results.</p> <p>iv. Continue with the next step.</p>
New Jira item	The test results of the test that you configure for a run are exported to a new Jira issue. The issue is created in your project in Jira after the test run is completed.	<p>Perform the following steps:</p> <p>i. Select the New Jira item option.</p> <p>ii. Continue with the next step.</p>
None	<p>The test results are not exported.</p> <p> Note: This is the default option.</p>	<p>Perform the following steps when:</p> <ul style="list-style-type: none"> ▪ You do not want to export the test results. ▪ You selected a Jira issue in a previous run of the same test, and you do not want to export results of the current run. <p>i. Select the None option.</p> <p>ii. Continue with the next step.</p>

16. Follow the instructions if you want to change the location of a Kubernetes cluster for running the test:

- a. Click the **LOCATION** tab, if it is not already open.

The **Default Cluster** is the default location where the test runs, and it is listed under the **Host** column.

The information about the availability of the default location is displayed.



Important: You must have added the following remote hosts to your project that are then displayed under the **Override** column:

- Docker hosts that are registered with Rational® Test Automation Server.
- Kubernetes clusters that are registered with a team space.



Notes:



- If the remote Docker hosts or clusters are not added to your project, the option **No override options** is displayed as the default value and the test runs in the Kubernetes cluster of Rational® Test Automation Server.
- If remote Docker hosts or clusters are added to your project, the added Docker hosts or clusters are displayed along with their availability status and ownership information.

b. Select the location where you want to run the test asset from the following options:

- Select the **Default Cluster** when no remote Docker hosts or clusters are available in your project.
- Select the remote Docker host or cluster from the list when a remote Docker host or cluster is available in your project.
- Select **No override options**, if you selected any remote Docker host or cluster and want to revert to the **Default Cluster** to run the test asset.

c. Click **Execute**.


The test run is initiated.

Results

You have started or scheduled a test run of a Compound Test that contains mobile tests or a Functional mobile test.

What to do next

You can choose to perform any of the following tasks:

- Stop the test run at any point after the test run is initiated from the **Execution** page. See [Stopping test runs on page 403](#).
- Cancel a scheduled test run from the **Execution** page. See [Canceling scheduled test runs on page 406](#).
- View all the states of the test asset by clicking the **Show in the Progress page** icon  for the test asset for which you started or scheduled the run. See [Viewing the state of test assets on page 394](#).
- View the progress of the test from the **Progress** page. See [Viewing the progress of running test assets on page 398](#).
- View the results, reports, and logs of the test from the **Results** page after the test completes the run. See [Test results on page 455](#).

Related information

[Resetting the configuration settings for a test run on page 402](#)

[Test run configurations on page 249](#)

Configuring a run of a Functional Windows test

After you added the test resources that you created in the desktop client to the project, you can configure a Functional Windows test to be run on IBM® Rational® Test Automation Server or a remote Kubernetes cluster.

Before you begin

You must have completed the following tasks:

- Created a Functional Windows test in Rational® Functional Tester and added the test asset to the project repository on IBM® Rational® Test Automation Server.
- Read [Considerations for using Jaeger traces in reports on page 205](#), if you want to configure a run and you want Jaeger to report the test results for the test.
- Read and completed the tasks mentioned in [Test run considerations for running tests on remote performance agents on page 211](#), if you want to use a remote agent as the location to run the test.
- Read and completed the tasks mentioned in [Considerations for using a remote Kubernetes cluster on page 235](#), if you want to use a remote Kubernetes cluster as a location to run the test.
- Completed the Xray integration, if you want to export the test results to a Jira issue in your Xray project in Jira. See [Integration with Xray test management for Jira on page 593](#).
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created a project in your team space in Rational® Test Automation Server. See [Adding a project on page 619](#).
- Added the remote Git repository that contains the test resources to the project. See [Adding repositories to a server project on page 620](#).

About this task

You can run the Functional Windows tests after you create the tests in Rational® Functional Tester, and then add them to your project on IBM® Rational® Test Automation Server.

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the test assets, and then click **Execution**.
3. Select the branch of the repository that contains the test assets.

The test assets that are contained in the selected branch of the repository are displayed in the following tabs on the **Execution** page:

Tab	Description
SUITES	Lists all suites, compound tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules that are in the selected branch of the repository.
TESTS	Lists all API tests, functional tests, or performance tests that are in the selected branch of the repository.
ADVANCED	Lists all assets that are displayed when custom filters are applied for assets in the selected branch of the repository.

4. Select the tab based on the type of test assets that you want to run as indicated in the following table:


If...	Then...
You want to run AFT Suites, API Suites, Compound Tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules,.	Click the SUITES tab.
You want to run API tests, functional tests, or performance tests.	Click the TESTS tab.
You want to find Suites and tests by using custom filters, and then run Suites and tests.	Click the ADVANCED tab.







The test assets in the selected tab are displayed.


5. Identify the test asset that you want to run by performing any of the following actions:































- Scroll through the list.


 **Tip:** You can hover over the icon in the **Type** column to know the type of the test asset.

 **Note:** You can also identify the type of the asset from the icon that represents the test type as shown in the following table:


Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the ADVANCED tab
	API test			
	Functional test			



Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the ADVANCED tab
	Performance test			
	AFT Suite			
	API Suite			
	Compound Test			
	HCL AppScan CodeSweep			
	JMeter Test			
	JUnit Test			
	Postman test			
	Rate Schedule			
	VU Schedule			

- Search for the test asset by entering any text contained in the test asset name in the **Search** field.
- Click the **Filter** icon  in the **SUITES** or **TESTS** tab to filter the displayed assets based on the asset type.

For example, select **API Suite** in the **SUITES** tab to display all API Suites or select **Functional Test** in the **TESTS** tab to display all functional tests that are in the selected branch of the repository.

- Click the **Filter** icon  in the **ADVANCED** tab, and then create a filter query by using the **New filter** option by performing the following steps:
 - Click **New filter**.
 - Enter a name for the filter.
 - Select an operator, and add a rule or a group of rules.
 - Add or enter the relevant parameters and either select or enter the condition and the criteria for the condition.

You can select a parameter from the following list:

- Type
 - Test Asset Name
 - Test Asset Path
 - Last Result
 - Next Run
 - Components
- e. Save the filter query to save and apply the filter query to filter the assets based on the query.

The test assets that match the filter criteria are displayed.

- Retrieve and apply a saved filter query, if you have saved filter queries previously by performing the following steps:



Note: The filter query applied previously is selected and the assets based on that filter query are displayed. To apply a different filter query, you must have created and saved the filter query.

- a. Click the **Filter** icon  in the **ADVANCED** tab.


The filter queries that you created and saved are displayed.

- b. Click the filter that you want to apply.

The test assets that match the filter criteria are displayed.

You have identified the test asset that you want to run.

Clicking the test name displays the **Details** panel. You can view the details of the test such as the description, the branch in the repository that contains the asset, the Git details, and the details of the commits to the repository. You can also view the history of the test runs for the specific test under the **History** tab of the **Details** panel.

6. Click the **Execute** icon  in the row of the identified test asset.

The **Execute test asset** dialog is displayed.



Notes:

- If you have configured some or all of the settings for the current test run, and you do not want to continue with those settings, you can reset the settings by clicking **Reset**.
- If you want to repeat a test run and do not want to use the saved settings from a previous run, you can reset all the saved settings to their default values by clicking **Reset**.

7. Select the version of the test resources that you want to run by completing any of the following actions:



Note: The test resources in the version can contain the test assets, datasets, AFT XML files, API environment tags, and other resources specific to projects created from any of the desktop clients.

- Expand the list in the **Version** field, find the version of the test resources, and then select the version.

Use the following details about the version of the test resources that are displayed to identify the version that you want:

- Commit message.
- Tags labeled by the user for the version committed.
- The user who committed the version to the repository.
- Relative time of the commit. For example, *2 hours ago* or *3 days ago*.

The list displays the versions of the test resources committed by all users to the branch in the repository. The versions are arranged with the latest version that is committed, and then followed by the versions committed previously.

- Expand the list in the **Version** field, and then search for the version that you want to select by entering a partial or the complete commit message of that version.

The version that matches the search criteria is displayed and it is selected for the test run.

The default value for the version selected for the run is the latest version in the selected branch of the repository. If you do not select any version, then the latest version is selected for the test run.



Notes:

- If you selected a version but you do not want to use that version in the test run, you can remove the selected version by clicking the ✕ icon. As a result, the default version is selected for the test run.
- If you repeated a test or ran the test again from the **Results** page, then the version of the test resources that you chose for the earlier run is shown as selected. You can either retain this version or select any other version from the list. You can also remove the previous version by clicking the ✕ icon.

8. Select the time for scheduling the test run from the following options:

- No action is required if you want to initiate the test run immediately after you click **Execute**.







Important: Click **Execute** only after you have configured the other settings in this dialog.



Note: The default time for scheduling a run is **Now**.

- Select **Schedule** and perform the following actions if you want to schedule a single test run or configure recurring test runs:


Schedule	Actions
<p>A single test run</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Select the Never option in the Repeat every field. Click Save. <p>The test run is scheduled to run at the selected time on the scheduled date.</p>
<p>Recurring test runs</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Set the frequency at which the test runs must run by entering the number in the Repeat every field, and then select the period from the list. You can select from the following options: <ul style="list-style-type: none"> ▪ Minute ▪ Hour ▪ Day ▪ Week ▪ Month <p>For example, if you want the test run to be run every day at the set time, enter 1 and, then select the Day(s) option.</p> <p>You can also schedule the test to run at the set time on specific weekdays by selecting the Week(s) option. For example, the following image displays the days selected for the test runs as <i>Monday, Wednesday, and Friday</i>:</p> <div data-bbox="708 1486 1421 1644" style="border: 1px solid black; padding: 5px; margin: 10px 0;"> <p>Repeat every <input style="width: 80px;" type="text" value="1"/> <input style="width: 150px;" type="text" value="Week(s)"/> </p> <p style="text-align: center;"> S M T W T F S </p> </div> Set the option to end the recurring test runs from the following options:

Schedule	Actions
	<ul style="list-style-type: none"> ▪ Select the Never option, if you do not want the recurring test runs to end. ▪ Select the On option, and then click the Calendar icon . You can select the date and time after which the scheduled test runs do not run. <p>f. Click Save.</p> <p>The recurring test runs are scheduled to start the first run at the selected time on the scheduled date, and to repeat the run at the frequency that you set.</p>

9. Enter a label for the test run that helps you to identify the test on the **Results** page.

After the test run completes, the label that you entered is displayed for the test under the **Labels** column on the **Results** page. After you have created a label, any member of the project can use that label.

The default value for the **Label** field is null or an empty field.

 **Important:** The configuration that you set for the test run in the **Execute test asset** dialog is preserved when you run the same test again. Those changes are not visible when another user logs in to Rational® Test Automation Server. For example, if you created new variables on the server, those variables are available only for you when the same test is run again.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

10. Click **Advanced** to make the following advanced configurations:

- a. Enter any JVM arguments that must be passed to the test run at run time in the **JVM Arguments** field, if applicable for the test.

For example, you can set a maximum Java heap size.

- b. Enter program arguments that must be passed to the test run at run time in the **Program Arguments**, if applicable for the test.

If the test that you want to configure supports the presentation of the test results data in different formats, then do the required tasks in the following scenarios:

If...	Then...
You want the test results data in different supported formats.	Enter <code>-history cisterna</code> in the Program Arguments field.

If...	Then...
You want to obtain the test results as a Jaeger trace.	Enter <code>-history jaeger</code> in the Program Arguments field.
You want the report as a test log and as a Jaeger trace.	Enter <code>-history jaeger, testlog</code> in the Program Arguments field.
Jaeger is already set as a program argument and you want to remove the Jaeger trace for your test.	Delete the <code>-history jaeger</code> entry in the Program Arguments field.



Note: The default report format is the *test log* format for the test reports.

- c. Enter the environment variables that must be passed to the test run at run time in the **Environment Variables** field, if applicable for the test.

For example, enter the environment variables when the third-party libraries that are used in the test run refer to the environment variables for configuration.



Note: You must separate the arguments or variables with a white space when you enter them in the same line or start each argument or variable on a new line.

The default value for each of the fields for the advanced settings is null or an empty field.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

11. Follow the instructions if you are running a test asset that contains datasets:

- a. Click the **DATA SOURCES** tab, if it is not already open.
- b. Consider the following information about datasets before you select a dataset:

The default value for the datasets in the **DATA SOURCES** tab is null if the test asset did not have an associated dataset. If the asset had an associated dataset, the default value is the associated dataset.

You can utilize the dataset stored as an Excel or CSV file to override the original dataset associated with the Suite, test, or schedule. For example, when you have associated a dataset in `.xlsx`, `.xls`, or `.csv` format with the test or schedule in desktop clients and if you have another set of data stored in an Excel or CSV file, then you can select that dataset from the **Override** list.



Remember: You must have uploaded the dataset as an Excel or CSV file into the Git repository, and ensured that both the original dataset (from the test asset) and new datasets (added to the project) have the same column names.

c. Select the dataset that you want to use in the test run from any of the following options:

- Select the dataset that is displayed as the default dataset when the test asset contains a single dataset.



Note: If there is only one dataset in the test asset, then that dataset is displayed as the default dataset.

- Select the dataset from the list.



Note: If there are multiple datasets in the test asset, the datasets are listed in their increasing alphabetical order.

- Select the dataset from the **Override** list to override the dataset that was associated with the test in the desktop client.




Important: If the test contains an encrypted dataset, the Project Owner must classify it in the **DATA SECURITY** tab on the Project page before you can select it. You must have added datasets to your project from the **Dataset** page for the datasets to be displayed in the **Override** list.


If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

12. Perform the following steps to provide the variables that must be passed to the test at the test run time. You can either enter the variables or import the file that contains the variables.

a. Click the **VARIABLES** tab, if it is not already open.

b. Choose one of the following methods to add the variables:

- To add new variables manually, click the **Add Variable** icon , enter the name, and value of the variable.
 - i. Enter `Windows_Device_Selection` as the name.
 - ii. Enter `<IP_address>` as the value, which is the IP address of the computer that contains the UI Test agent.


- To add new variables from your local computer or from the Git repository that is associated with your server project, click the **Upload** icon  and select the **Upload from local system** or **Browse from server** to select the variable file.



Note: You must have created a file with the variables before you can select the file.

13. Follow the instructions if you want to export the test results to a Jira issue in your Xray project in Jira:

- a. Click the **RESULTS** tab.
- b. Perform the actions listed in the following table to select an option to export the test results:

Export options	Results in...	Actions
Jira Xray	<p>The test results of the test that you configure for a run are exported to the selected issue after the test run is completed. The results are exported to the selected issue that exists in your Xray project in Jira.</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> i. Select the Jira Xray option. <p>The Select issue field is displayed.</p> <ol style="list-style-type: none"> ii. Click the field to view the list of issues in the Xray project in Jira. Alternatively, you can enter the issue key in the Jira Xray project. iii. Select the issue to which you want to export the test results. iv. Continue with the next step.
New Jira item	<p>The test results of the test that you configure for a run are exported to a new Jira issue. The issue is created in your project in Jira after the test run is completed.</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> i. Select the New Jira item option. ii. Continue with the next step.
None	<p>The test results are not exported.</p> <p> Note: This is the default option.</p>	<p>Perform the following steps when:</p> <ul style="list-style-type: none"> ▪ You do not want to export the test results. ▪ You selected a Jira issue in a previous run of the same test, and you do not want to export results of the current run. <ol style="list-style-type: none"> i. Select the None option. ii. Continue with the next step.

14. Select any of the following steps that depends on the location to run the test:

- If you want to run the tests on the remote agents, go to [Step 15 on page 329](#).
 - If you want to run the tests on a Kubernetes cluster, go to [Step 16 on page 330](#).
15. Follow the instructions if you want to run a test on the remote performance agent that is connected either to the default Kubernetes cluster or the remote Kubernetes cluster:

- a. Click the **LOCATION** tab, if it is not already open.

The performance agents that are configured in the test asset are listed under the **Host** column. The information about the availability of the performance agent is displayed.



Note: You must have added performance agents to your project from the **Agents and Intercepts** page for the performance agents to be displayed under the **Override** column.

The default value for the performance agents is null or an empty field if no performance agents were configured in the test asset. If the test asset contains performance agents that are configured, then the default performance agent is the first item to be displayed on the list of performance agents listed in the increasing alphabetical order.

- b. Select the performance agent where you want to run the test asset in the following scenarios:

If...	Then...	Action
The performance agent that is specified in the test assets is available and no other performance agents are added to the project.	The performance agent that is specified in the test assets is displayed in the Override column.	Select the performance agent in the Host that is also displayed in the Override column as the location to run the test.
The performance agent that is specified in the test assets is not available and no other performance agents are added to the project.	There is no performance agent displayed for override in the Override column.	You cannot run the test on the performance agent. You must wait until the performance agent is available to run the test on the agent.
The agent that is specified in the test assets is available and there are other agents that are added to the project.	Performance agents that have capabilities that match with the agent capabilities specified in the test assets are listed in the Override column as follows: <ul style="list-style-type: none"> ▪ Performance agents with capabilities that best match the capabil- 	Perform any of the following actions: <ul style="list-style-type: none"> ▪ Select the performance agent in the Host that is also displayed in the Override column as the location to run the test.

If...	Then...	Action
	<p>ities in the agent in the asset are at the top of the list.</p> <ul style="list-style-type: none"> Performance agents with capabilities that do not match with the capabilities in the agent in the asset are listed subsequently. <p>You can find details of both performance agents for the matching capabilities when you hover over the performance agent in the Override column.</p>	<ul style="list-style-type: none"> Select a performance agent in the Override column as the location to run the test.


c. Click **Execute**.

The test run is initiated.

16. Follow the instructions if you want to change the location of a Kubernetes cluster for running the test:

a. Click the **LOCATION** tab, if it is not already open.

The **Default Cluster** is the default location where the test runs, and it is listed under the **Host** column. The information about the availability of the default location is displayed.

 **Important:** You must have added the following remote hosts to your project that are then displayed under the **Override** column:

- Docker hosts that are registered with Rational® Test Automation Server.
- Kubernetes clusters that are registered with a team space.

 **Notes:**

- If the remote Docker hosts or clusters are not added to your project, the option **No override options** is displayed as the default value and the test runs in the Kubernetes cluster of Rational® Test Automation Server.
- If remote Docker hosts or clusters are added to your project, the added Docker hosts or clusters are displayed along with their availability status and ownership information.

b. Select the location where you want to run the test asset from the following options:

- Select the **Default Cluster** when no remote Docker hosts or clusters are available in your project.
- Select the remote Docker host or cluster from the list when a remote Docker host or cluster is available in your project.
- Select **No override options**, if you selected any remote Docker host or cluster and want to revert to the **Default Cluster** to run the test asset.

c. Click **Execute**.


The test run is initiated.

Results

You have started or scheduled a test run of a Functional Windows test.

What to do next

You can choose to perform any of the following tasks:

- Stop the test run at any point after the test run is initiated from the **Execution** page. See [Stopping test runs on page 403](#).
- Cancel a scheduled test run from the **Execution** page. See [Canceling scheduled test runs on page 406](#).
- View all the states of the test asset by clicking the **Show in the Progress page** icon  for the test asset for which you started or scheduled the run. See [Viewing the state of test assets on page 394](#).
- View the progress of the test from the **Progress** page. See [Viewing the progress of running test assets on page 398](#).
- View the results, reports, and logs of the test from the **Results** page after the test completes the run. See [Test results on page 455](#).

Related information

[Resetting the configuration settings for a test run on page 402](#)

[Test run configurations on page 249](#)

Configuring a run of a Compound Test that contains performance tests or a Performance test

After you added the test resources that you created in the desktop client to the project, you can configure a Compound Test that contains performance tests or a Performance test to be run on IBM® Rational® Test Automation Server or a remote Kubernetes cluster.

Before you begin

You must have completed the following tasks:

- Created performance tests under a Compound Test or a Performance test in Rational® Performance Tester.
- Read and completed the tasks mentioned in [Test run considerations for running tests on remote performance agents on page 211](#), if you want to use a remote agent as the location to run the test.
- Read and completed the tasks mentioned in [Considerations for using a remote Kubernetes cluster on page 235](#), if you want to use a remote Kubernetes cluster as a location to run the test.
- Completed the Xray integration, if you want to export the test results to a Jira issue in your Xray project in Jira. See [Integration with Xray test management for Jira on page 593](#).
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created a project in your team space in Rational® Test Automation Server. See [Adding a project on page 619](#).
- Added the remote Git repository that contains the test resources to the project. See [Adding repositories to a server project on page 620](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the test assets, and then click **Execution**.
3. Select the branch of the repository that contains the test assets.

The test assets that are contained in the selected branch of the repository are displayed in the following tabs on the **Execution** page:

Tab	Description
SUITES	Lists all suites, compound tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules that are in the selected branch of the repository.
TESTS	Lists all API tests, functional tests, or performance tests that are in the selected branch of the repository.
ADVANCED	Lists all assets that are displayed when custom filters are applied for assets in the selected branch of the repository.

4. Select the tab based on the type of test assets that you want to run as indicated in the following table:

If...	Then...
You want to run AFT Suites, API Suites, Compound Tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules,.	Click the SUITES tab.


If...	Then...
You want to run API tests, functional tests, or performance tests.	Click the TESTS tab.
You want to find Suites and tests by using custom filters, and then run Suites and tests.	Click the ADVANCED tab.




























The test assets in the selected tab are displayed.


5. Identify the test asset that you want to run by performing any of the following actions:










- Scroll through the list.


 **Tip:** You can hover over the icon in the **Type** column to know the type of the test asset.

 **Note:** You can also identify the type of the asset from the icon that represents the test type as shown in the following table:


Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the ADVANCED tab
	API test			
	Functional test			
	Performance test			
	AFT Suite			
	API Suite			
	Compound Test			
	HCL AppScan CodeSweep			
	JMeter Test			
	JUnit Test			



Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the AD-VANCED tab
	Postman test			
	Rate Schedule			
	VU Schedule			

- Search for the test asset by entering any text contained in the test asset name in the **Search** field.
- Click the **Filter** icon  in the **SUITES** or **TESTS** tab to filter the displayed assets based on the asset type.

For example, select **API Suite** in the **SUITES** tab to display all API Suites or select **Functional Test** in the **TESTS** tab to display all functional tests that are in the selected branch of the repository.


- Click the **Filter** icon  in the **ADVANCED** tab, and then create a filter query by using the **New filter** option by performing the following steps:
 - Click **New filter**.
 - Enter a name for the filter.
 - Select an operator, and add a rule or a group of rules.
 - Add or enter the relevant parameters and either select or enter the condition and the criteria for the condition.

You can select a parameter from the following list:

- Type
 - Test Asset Name
 - Test Asset Path
 - Last Result
 - Next Run
 - Components
- Save the filter query to save and apply the filter query to filter the assets based on the query.

The test assets that match the filter criteria are displayed.

- Retrieve and apply a saved filter query, if you have saved filter queries previously by performing the following steps:

 **Note:** The filter query applied previously is selected and the assets based on that filter query are displayed. To apply a different filter query, you must have created and saved the filter query.

- a. Click the **Filter** icon  in the **ADVANCED** tab.


The filter queries that you created and saved are displayed.

- b. Click the filter that you want to apply.

The test assets that match the filter criteria are displayed.

You have identified the test asset that you want to run.

Clicking the test name displays the **Details** panel. You can view the details of the test such as the description, the branch in the repository that contains the asset, the Git details, and the details of the commits to the repository. You can also view the history of the test runs for the specific test under the **History** tab of the **Details** panel.


6. Click the **Execute** icon  in the row of the identified test asset.

The **Execute test asset** dialog is displayed.

 **Notes:**

- If you have configured some or all of the settings for the current test run, and you do not want to continue with those settings, you can reset the settings by clicking **Reset**.
- If you want to repeat a test run and do not want to use the saved settings from a previous run, you can reset all the saved settings to their default values by clicking **Reset**.

7. Select the version of the test resources that you want to run by completing any of the following actions:

 **Note:** The test resources in the version can contain the test assets, datasets, AFT XML files, API environment tags, and other resources specific to projects created from any of the desktop clients.

- Expand the list in the **Version** field, find the version of the test resources, and then select the version.

Use the following details about the version of the test resources that are displayed to identify the version that you want:

- Commit message.
- Tags labeled by the user for the version committed.
- The user who committed the version to the repository.
- Relative time of the commit. For example, *2 hours ago* or *3 days ago*.

The list displays the versions of the test resources committed by all users to the branch in the repository. The versions are arranged with the latest version that is committed, and then followed by the versions committed previously.

- Expand the list in the **Version** field, and then search for the version that you want to select by entering a partial or the complete commit message of that version.

The version that matches the search criteria is displayed and it is selected for the test run.

The default value for the version selected for the run is the latest version in the selected branch of the repository. If you do not select any version, then the latest version is selected for the test run.



Notes:

- If you selected a version but you do not want to use that version in the test run, you can remove the selected version by clicking the ✕ icon. As a result, the default version is selected for the test run.
- If you repeated a test or ran the test again from the **Results** page, then the version of the test resources that you chose for the earlier run is shown as selected. You can either retain this version or select any other version from the list. You can also remove the previous version by clicking the ✕ icon.

8. Select the time for scheduling the test run from the following options:

- No action is required if you want to initiate the test run immediately after you click **Execute**.






Important: Click **Execute** only after you have configured the other settings in this dialog.



Note: The default time for scheduling a run is **Now**.

- Select **Schedule** and perform the following actions if you want to schedule a single test run or configure recurring test runs:


Schedule	Actions
A single test run	Perform the following steps: <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Select the Never option in the Repeat every field. Click Save.

Schedule	Actions
	The test run is scheduled to run at the selected time on the scheduled date.
Recurring test runs	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Set the frequency at which the test runs must run by entering the number in the Repeat every field, and then select the period from the list. You can select from the following options: <ul style="list-style-type: none"> Minute Hour Day Week Month <p>For example, if you want the test run to be run every day at the set time, enter 1 and, then select the Day(s) option.</p> <p>You can also schedule the test to run at the set time on specific weekdays by selecting the Week(s) option. For example, the following image displays the days selected for the test runs as <i>Monday, Wednesday, and Friday</i>:</p> <div data-bbox="708 1178 1422 1339" data-label="Form"> </div> Set the option to end the recurring test runs from the following options: <ul style="list-style-type: none"> Select the Never option, if you do not want the recurring test runs to end. Select the On option, and then click the Calendar icon . You can select the date and time after which the scheduled test runs do not run. Click Save. <p>The recurring test runs are scheduled to start the first run at the selected time on the scheduled date, and to repeat the run at the frequency that you set.</p>

9. Enter a label for the test run that helps you to identify the test on the **Results** page.

After the test run completes, the label that you entered is displayed for the test under the **Labels** column on the **Results** page. After you have created a label, any member of the project can use that label.

The default value for the **Label** field is null or an empty field.

 **Important:** The configuration that you set for the test run in the **Execute test asset** dialog is preserved when you run the same test again. Those changes are not visible when another user logs in to Rational® Test Automation Server. For example, if you created new variables on the server, those variables are available only for you when the same test is run again.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

10. Click **Advanced** to make the following advanced configurations:

a. Enter any JVM arguments that must be passed to the test run at run time in the **JVM Arguments** field, if applicable for the test.

For example, you can set a maximum Java heap size.

b. Enter program arguments that must be passed to the test run at run time in the **Program Arguments**, if applicable for the test.

If the test that you want to configure supports the presentation of the test results data in different formats, then do the required tasks in the following scenarios:

If...	Then...
You want the test results data in different supported formats.	Enter <code>-history cisterna</code> in the Program Arguments field.
You want to obtain the test results as a Jaeger trace.	Enter <code>-history jaeger</code> in the Program Arguments field.
You want the report as a test log and as a Jaeger trace.	Enter <code>-history jaeger, testlog</code> in the Program Arguments field.
Jaeger is already set as a program argument and you want to remove the Jaeger trace for your test.	Delete the <code>-history jaeger</code> entry in the Program Arguments field.



Note: The default report format is the *test log* format for the test reports.

- c. Enter the environment variables that must be passed to the test run at run time in the **Environment Variables** field, if applicable for the test.

For example, enter the environment variables when the third-party libraries that are used in the test run refer to the environment variables for configuration.



Note: You must separate the arguments or variables with a white space when you enter them in the same line or start each argument or variable on a new line.

The default value for each of the fields for the advanced settings is null or an empty field.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

11. Follow the instructions if you are running a test asset that contains datasets:

- a. Click the **DATA SOURCES** tab, if it is not already open.
- b. Consider the following information about datasets before you select a dataset:

The default value for the datasets in the **DATA SOURCES** tab is null if the test asset did not have an associated dataset. If the asset had an associated dataset, the default value is the associated dataset.

You can utilize the dataset stored as an Excel or CSV file to override the original dataset associated with the Suite, test, or schedule. For example, when you have associated a dataset in `.xlsx`, `.xls`, or `.csv` format with the test or schedule in desktop clients and if you have another set of data stored in an Excel or CSV file, then you can select that dataset from the **Override** list.



Remember: You must have uploaded the dataset as an Excel or CSV file into the Git repository, and ensured that both the original dataset (from the test asset) and new datasets (added to the project) have the same column names.

- c. Select the dataset that you want to use in the test run from any of the following options:
 - Select the dataset that is displayed as the default dataset when the test asset contains a single dataset.



Note: If there is only one dataset in the test asset, then that dataset is displayed as the default dataset.

- Select the dataset from the list.



Note: If there are multiple datasets in the test asset, the datasets are listed in their increasing alphabetical order.

- Select the dataset from the **Override** list to override the dataset that was associated with the test in the desktop client.





Important: If the test contains an encrypted dataset, the Project Owner must classify it in the **DATA SECURITY** tab on the Project page before you can select it. You must have added datasets to your project from the **Dataset** page for the datasets to be displayed in the **Override** list.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

12. Follow the instructions if the test requires a variable that must be passed to the test at the test run time.

a. Click the **VARIABLES** tab, if it is not already open.

b. Choose one of the following methods to add the variables:

- To add new variables manually, click the **Add Variable** icon , enter the name, and value of the variable.
- To add new variables from your local computer or from the Git repository that is associated with your server project, click the **Upload** icon  and select the **Upload from local system** or **Browse from server** to select the variable file.



Note: You must have created a file with the variables before you can select the file.

The default value for the variables is null or an empty field.


If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

13. Follow the instructions if you want to export the test results to a Jira issue in your Xray project in Jira:

a. Click the **RESULTS** tab.


b. Perform the actions listed in the following table to select an option to export the test results:

Export options	Results in...	Actions
Jira Xray	The test results of the test that you configure for a run are exported to the selected issue after the test run is completed. The results are exported to the	Perform the following steps: i. Select the Jira Xray option. The Select issue field is displayed.

Export options	Results in...	Actions
	selected issue that exists in your Xray project in Jira.	<ul style="list-style-type: none"> ii. Click the field to view the list of issues in the Xray project in Jira. Alternatively, you can enter the issue key in the Jira Xray project. iii. Select the issue to which you want to export the test results. iv. Continue with the next step.
New Jira item	The test results of the test that you configure for a run are exported to a new Jira issue. The issue is created in your project in Jira after the test run is completed.	Perform the following steps: <ul style="list-style-type: none"> i. Select the New Jira item option. ii. Continue with the next step.
None	The test results are not exported.  Note: This is the default option.	Perform the following steps when: <ul style="list-style-type: none"> ▪ You do not want to export the test results. ▪ You selected a Jira issue in a previous run of the same test, and you do not want to export results of the current run. <ul style="list-style-type: none"> i. Select the None option. ii. Continue with the next step.

14. Select any of the following steps that depends on the location to run the test:
 - If you want to run the tests on the remote agents, go to [Step 15 on page 341](#).
 - If you want to run the tests on a Kubernetes cluster, go to [Step 16 on page 343](#).
15. Follow the instructions if you want to run a test on the remote performance agent that is connected either to the default Kubernetes cluster or the remote Kubernetes cluster:
 - a. Click the **LOCATION** tab, if it is not already open.

The performance agents that are configured in the test asset are listed under the **Host** column. The information about the availability of the performance agent is displayed.

 **Note:** You must have added performance agents to your project from the **Agents and Intercepts** page for the performance agents to be displayed under the **Override** column.

The default value for the performance agents is null or an empty field if no performance agents were configured in the test asset. If the test asset contains performance agents that are configured, then the default performance agent is the first item to be displayed on the list of performance agents listed in the increasing alphabetical order.

b. Select the performance agent where you want to run the test asset in the following scenarios:

If...	Then...	Action
<p>The performance agent that is specified in the test assets is available and no other performance agents are added to the project.</p>	<p>The performance agent that is specified in the test assets is displayed in the Override column.</p>	<p>Select the performance agent in the Host that is also displayed in the Override column as the location to run the test.</p>
<p>The performance agent that is specified in the test assets is not available and no other performance agents are added to the project.</p>	<p>There is no performance agent displayed for override in the Override column.</p>	<p>You cannot run the test on the performance agent.</p> <p>You must wait until the performance agent is available to run the test on the agent.</p>
<p>The agent that is specified in the test assets is available and there are other agents that are added to the project.</p>	<p>Performance agents that have capabilities that match with the agent capabilities specified in the test assets are listed in the Override column as follows:</p> <ul style="list-style-type: none"> ▪ Performance agents with capabilities that best match the capabilities in the agent in the asset are at the top of the list. ▪ Performance agents with capabilities that do not match with the capabilities in the agent in the asset are listed subsequently. <p>You can find details of both performance agents for the matching capabilities when you hover over the performance agent in the Override column.</p>	<p>Perform any of the following actions:</p> <ul style="list-style-type: none"> ▪ Select the performance agent in the Host that is also displayed in the Override column as the location to run the test. ▪ Select a performance agent in the Override column as the location to run the test.

c. Click **Execute**.

The test run is initiated.

16. Follow the instructions if you want to change the location of a Kubernetes cluster for running the test:

a. Click the **LOCATION** tab, if it is not already open.

The **Default Cluster** is the default location where the test runs, and it is listed under the **Host** column. The information about the availability of the default location is displayed.



Important: You must have added the following remote hosts to your project that are then displayed under the **Override** column:

- Docker hosts that are registered with Rational® Test Automation Server.
- Kubernetes clusters that are registered with a team space.



Notes:

- If the remote Docker hosts or clusters are not added to your project, the option **No override options** is displayed as the default value and the test runs in the Kubernetes cluster of Rational® Test Automation Server.
- If remote Docker hosts or clusters are added to your project, the added Docker hosts or clusters are displayed along with their availability status and ownership information.

b. Select the location where you want to run the test asset from the following options:

- Select the **Default Cluster** when no remote Docker hosts or clusters are available in your project.
- Select the remote Docker host or cluster from the list when a remote Docker host or cluster is available in your project.
- Select **No override options**, if you selected any remote Docker host or cluster and want to revert to the **Default Cluster** to run the test asset.

c. Click **Execute**.


The test run is initiated.

Results

You have started or scheduled a test run of a Compound Test that contains performance tests or a Performance test.

What to do next

You can choose to perform any of the following tasks:

- Stop the test run at any point after the test run is initiated from the **Execution** page. See [Stopping test runs on page 403](#).
- Cancel a scheduled test run from the **Execution** page. See [Canceling scheduled test runs on page 406](#).
- View all the states of the test asset by clicking the **Show in the Progress page** icon  for the test asset for which you started or scheduled the run. See [Viewing the state of test assets on page 394](#).

- View the progress of the test from the **Progress** page. See [Viewing the progress of running test assets on page 398](#).
- View the results, reports, and logs of the test from the **Results** page after the test completes the run. See [Test results on page 455](#).

Related information

[Resetting the configuration settings for a test run on page 402](#)

[Test run configurations on page 249](#)

Configuring a run of a JMeter test

After you added the test resources that you created in JMeter to the server project, you can configure a JMeter test to be run on IBM® Rational® Test Automation Server or a remote Kubernetes cluster.

Before you begin

You must have completed the following tasks:

- Read and completed the tasks mentioned in [Test run considerations for JMeter tests on page 205](#).
- Read and completed the tasks mentioned in [Considerations for using a remote Kubernetes cluster on page 235](#), if you want to use a remote Kubernetes cluster as a location to run the test.
- Completed the Xray integration, if you want to export the test results to a Jira issue in your Xray project in Jira. See [Integration with Xray test management for Jira on page 593](#).
- Ensured that the JMeter extension on IBM® Rational® Test Automation Server is enabled before you configure a test run.
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created a project in your team space in Rational® Test Automation Server. See [Adding a project on page 619](#).
- Added the remote Git repository that contains the test resources to the project. See [Adding repositories to a server project on page 620](#).

About this task



Important: You can run the JMeter tests only on the default cluster of IBM® Rational® Test Automation Server or a remote Kubernetes cluster. You cannot run the JMeter tests on remote agents or remote Docker hosts that are registered with IBM® Rational® Test Automation Server.

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the test assets, and then click **Execution**.
3. Select the branch of the repository that contains the test assets.

The test assets that are contained in the selected branch of the repository are displayed in the following tabs on the **Execution** page:

Tab	Description
SUITES	Lists all suites, compound tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules that are in the selected branch of the repository.
TESTS	Lists all API tests, functional tests, or performance tests that are in the selected branch of the repository.
ADVANCED	Lists all assets that are displayed when custom filters are applied for assets in the selected branch of the repository.

4. Select the tab based on the type of test assets that you want to run as indicated in the following table:


If...	Then...
You want to run AFT Suites, API Suites, Compound Tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules,.	Click the SUITES tab.
You want to run API tests, functional tests, or performance tests.	Click the TESTS tab.
You want to find Suites and tests by using custom filters, and then run Suites and tests.	Click the ADVANCED tab.





































The test assets in the selected tab are displayed.


5. Identify the test asset that you want to run by performing any of the following actions:

- Scroll through the list.


 **Tip:** You can hover over the icon in the **Type** column to know the type of the test asset.

 **Note:** You can also identify the type of the asset from the icon that represents the test type as shown in the following table:

Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the AD-VANCED tab
	API test			
	Functional test			
	Performance test			
	AFT Suite			
	API Suite			
	Compound Test			
	HCL AppScan CodeSweep			
	JMeter Test			
	JUnit Test			
	Postman test			
	Rate Schedule			
	VU Schedule			

- Search for the test asset by entering any text contained in the test asset name in the **Search** field.
- Click the **Filter** icon  in the **SUITES** or **TESTS** tab to filter the displayed assets based on the asset type.

For example, select **API Suite** in the **SUITES** tab to display all API Suites or select **Functional Test** in the **TESTS** tab to display all functional tests that are in the selected branch of the repository.

- Click the **Filter** icon  in the **ADVANCED** tab, and then create a filter query by using the **New filter** option by performing the following steps:
 - a. Click **New filter**.
 - b. Enter a name for the filter.
 - c. Select an operator, and add a rule or a group of rules.
 - d. Add or enter the relevant parameters and either select or enter the condition and the criteria for the condition.

You can select a parameter from the following list:

- Type
 - Test Asset Name
 - Test Asset Path
 - Last Result
 - Next Run
 - Components
- e. Save the filter query to save and apply the filter query to filter the assets based on the query.

The test assets that match the filter criteria are displayed.

- Retrieve and apply a saved filter query, if you have saved filter queries previously by performing the following steps:



Note: The filter query applied previously is selected and the assets based on that filter query are displayed. To apply a different filter query, you must have created and saved the filter query.

- a. Click the **Filter** icon  in the **ADVANCED** tab.


The filter queries that you created and saved are displayed.

- b. Click the filter that you want to apply.

The test assets that match the filter criteria are displayed.

You have identified the test asset that you want to run.

Clicking the test name displays the **Details** panel. You can view the details of the test such as the description, the branch in the repository that contains the asset, the Git details, and the details of the commits to the repository. You can also view the history of the test runs for the specific test under the **History** tab of the **Details** panel.

6. Click the **Execute** icon  in the row of the identified test asset.

The **Execute test asset** dialog is displayed.



Notes:

- If you have configured some or all of the settings for the current test run, and you do not want to continue with those settings, you can reset the settings by clicking **Reset**.
- If you want to repeat a test run and do not want to use the saved settings from a previous run, you can reset all the saved settings to their default values by clicking **Reset**.

7. Select the version of the test resources that you want to run by completing any of the following actions:

- Expand the list in the **Version** field, find the version of the test resources, and then select the version.

Use the following details about the version of the test resources that are displayed to identify the version that you want:

- Commit message.
- Tags labeled by the user for the version committed.
- The user who committed the version to the repository.
- Relative time of the commit. For example, *2 hours ago* or *3 days ago*.

The list displays the versions of the test resources committed by all users to the branch in the repository. The versions are arranged with the latest version that is committed, and then followed by the versions committed previously.

- Expand the list in the **Version** field, and then search for the version that you want to select by entering a partial or the complete commit message of that version.

The version that matches the search criteria is displayed and it is selected for the test run.

8. Select the time for scheduling the test run from the following options:

- No action is required if you want to initiate the test run immediately after you click **Execute**.





Important: Click **Execute** only after you have configured the other settings in this dialog.




Note: The default time for scheduling a run is **Now**.

- Select **Schedule** and perform the following actions if you want to schedule a single test run or configure recurring test runs:

Schedule	Actions
A single test run	Perform the following steps:


Schedule	Actions
	<p>a. Click the Calendar icon  in the row of the Start field.</p> <p>b. Select the date.</p> <p>c. Select the time at which the test run must start.</p> <p>d. Select the Never option in the Repeat every field.</p> <p>e. Click Save.</p> <p>The test run is scheduled to run at the selected time on the scheduled date.</p>
Recurring test runs	<p>Perform the following steps:</p> <p>a. Click the Calendar icon  in the row of the Start field.</p> <p>b. Select the date.</p> <p>c. Select the time at which the test run must start.</p> <p>d. Set the frequency at which the test runs must run by entering the number in the Repeat every field, and then select the period from the list. You can select from the following options:</p> <ul style="list-style-type: none"> ▪ Minute ▪ Hour ▪ Day ▪ Week ▪ Month <p>For example, if you want the test run to be run every day at the set time, enter 1 and, then select the Day(s) option.</p> <p>You can also schedule the test to run at the set time on specific weekdays by selecting the Week(s) option. For example, the following image displays the days selected for the test runs as <i>Monday, Wednesday, and Friday</i>:</p> <div data-bbox="708 1413 1421 1572" style="border: 1px solid black; padding: 5px; margin: 10px 0;"> <p>Repeat every <input style="width: 60px;" type="text" value="1"/> <input style="width: 100px;" type="text" value="Week(s)"/> ▾</p> <p style="text-align: center;"> <input type="radio"/> S <input checked="" type="radio"/> M <input type="radio"/> T <input checked="" type="radio"/> W <input type="radio"/> T <input checked="" type="radio"/> F <input type="radio"/> S </p> </div> <p>e. Set the option to end the recurring test runs from the following options:</p>

Schedule	Actions
	<ul style="list-style-type: none"> ▪ Select the Never option, if you do not want the recurring test runs to end. ▪ Select the On option, and then click the Calendar icon . You can select the date and time after which the scheduled test runs do not run. <p>f. Click Save.</p> <p>The recurring test runs are scheduled to start the first run at the selected time on the scheduled date, and to repeat the run at the frequency that you set.</p>

9. Enter a label for the test run that helps you to identify the test on the **Results** page.

After the test run completes, the label that you entered is displayed for the test under the **Labels** column on the **Results** page. After you have created a label, any member of the project can use that label.

The default value for the **Label** field is null or an empty field.

 **Important:** The configuration that you set for the test run in the **Execute test asset** dialog is preserved when you run the same test again. Those changes are not visible when another user logs in to Rational® Test Automation Server. For example, if you created new variables on the server, those variables are available only for you when the same test is run again.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.


10. Click **Advanced** to make the following advanced configurations:

a. Enter any JVM arguments that must be passed to the test run at run time in the **JVM Arguments** field, if applicable for the test.

For example, you can set a maximum Java heap size.

b. Enter program arguments that must be passed to the test run at run time in the **Program Arguments**, if applicable for the test.

For example, enter **-J** to override any JMeter property that is defined in the properties file.


 **Note:** You must separate the arguments or variables with a white space when you enter them in the same line or start each argument or variable on a new line.

The default value for each of the fields for the advanced settings is null or an empty field.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

11. Follow the instructions if you want to export the test results to a Jira issue in your Xray project in Jira:


- a. Click the **RESULTS** tab.
- b. Perform the actions listed in the following table to select an option to export the test results:

Export options	Results in...	Actions
Jira Xray	The test results of the test that you configure for a run are exported to the selected issue after the test run is completed. The results are exported to the selected issue that exists in your Xray project in Jira.	Perform the following steps: <ol style="list-style-type: none"> i. Select the Jira Xray option. <p>The Select issue field is displayed.</p> <ol style="list-style-type: none"> ii. Click the field to view the list of issues in the Xray project in Jira. Alternatively, you can enter the issue key in the Jira Xray project. iii. Select the issue to which you want to export the test results. iv. Continue with the next step.
New Jira item	The test results of the test that you configure for a run are exported to a new Jira issue. The issue is created in your project in Jira after the test run is completed.	Perform the following steps: <ol style="list-style-type: none"> i. Select the New Jira item option. ii. Continue with the next step.
None	The test results are not exported.  Note: This is the default option.	Perform the following steps when: <ul style="list-style-type: none"> ▪ You do not want to export the test results. ▪ You selected a Jira issue in a previous run of the same test, and you do not want to export results of the current run. <ol style="list-style-type: none"> i. Select the None option. ii. Continue with the next step.

12. Follow the instructions if you want to change the location of a Kubernetes cluster for running the test:

- a. Click the **LOCATION** tab, if it is not already open.

The **Default Cluster** is the default location where the test runs, and it is listed under the **Host** column. The information about the availability of the default location is displayed.

 **Important:** You must have added the registered remote Kubernetes clusters to your project that are then displayed under the **Override** column:



Notes:

- If the remote clusters are not added to your project, the option **No override options** is displayed as the default value and the test runs in the Kubernetes cluster of Rational® Test Automation Server.
- If remote clusters are added to your project, the added clusters are displayed along with their availability status and ownership information.

b. Select the location where you want to run the test asset from the following options:

- Select the **Default Cluster** when no remote clusters are available in your project.
- Select the remote cluster from the list when a remote cluster is available in your project.
- Select **No override options**, if you selected any remote cluster and want to revert to the **Default Cluster** to run the test asset.

c. Click **Execute**.


The test run is initiated.

Results

You have started or scheduled a test run of a JMeter test.

What to do next

You can choose to perform any of the following tasks:

- Stop the test run at any point after the test run is initiated from the **Execution** page. See [Stopping test runs on page 403](#).
- Cancel a scheduled test run from the **Execution** page. See [Canceling scheduled test runs on page 406](#).
- View all the states of the test asset by clicking the **Show in the Progress page** icon  for the test asset for which you started or scheduled the run. See [Viewing the state of test assets on page 394](#).
- View the progress of the test from the **Progress** page. See [Viewing the progress of running test assets on page 398](#).
- View the results, reports, and logs of the test from the **Results** page after the test completes the run. See [Test results on page 455](#).

Related information

[Resetting the configuration settings for a test run on page 402](#)

[Test run configurations on page 249](#)

Configuring a run of a JUnit test

After you added the test resources that contain the Maven project with the JUnit tests to the server project, you can configure a JUnit test run on IBM® Rational® Test Automation Server or a remote Kubernetes cluster.

Before you begin

You must have completed the following tasks:

- Read and completed the tasks mentioned in [Test run considerations for JUnit tests on page 207](#) before you configure a JUnit test run.
- Read and completed the tasks mentioned in [Considerations for using a remote Kubernetes cluster on page 235](#), if you want to use a remote Kubernetes cluster as a location to run the test.
- Completed the Xray integration, if you want to export the test results to a Jira issue in your Xray project in Jira. See [Integration with Xray test management for Jira on page 593](#).
- Ensured that the JUnit extension on IBM® Rational® Test Automation Server is enabled before you configure a test run.
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created a project in your team space in Rational® Test Automation Server. See [Adding a project on page 619](#).
- Added the remote Git repository that contains the test resources to the project. See [Adding repositories to a server project on page 620](#).

About this task



Important: You can run the JUnit tests only on the default cluster of IBM® Rational® Test Automation Server or a remote Kubernetes cluster. You cannot run the JUnit tests on remote agents or remote Docker hosts that are registered with IBM® Rational® Test Automation Server.

After the test runs, you can view the results of the JUnit test in the following formats:

- Surefire
- Jaeger trace
- JUnit XML Report

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the test assets, and then click **Execution**.
3. Select the branch of the repository that contains the test assets.

The test assets that are contained in the selected branch of the repository are displayed in the following tabs on the **Execution** page:

Tab	Description
SUITES	Lists all suites, compound tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules that are in the selected branch of the repository.
TESTS	Lists all API tests, functional tests, or performance tests that are in the selected branch of the repository.
ADVANCED	Lists all assets that are displayed when custom filters are applied for assets in the selected branch of the repository.

4. Select the tab based on the type of test assets that you want to run as indicated in the following table:


If...	Then...
You want to run AFT Suites, API Suites, Compound Tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules,.	Click the SUITES tab.
You want to run API tests, functional tests, or performance tests.	Click the TESTS tab.
You want to find Suites and tests by using custom filters, and then run Suites and tests.	Click the ADVANCED tab.


The test assets in the selected tab are displayed.





































5. Identify the test asset that you want to run by performing any of the following actions:


- Scroll through the list.

 **Tip:** You can hover over the icon in the **Type** column to know the type of the test asset.


 **Note:** You can also identify the type of the asset from the icon that represents the test type as shown in the following table:



Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the ADVANCED tab
	API test			
	Functional test			
	Performance test			
	AFT Suite			
	API Suite			
	Compound Test			
	HCL AppScan CodeSweep			
	JMeter Test			
	JUnit Test			
	Postman test			
	Rate Schedule			
	VU Schedule			

- Search for the test asset by entering any text contained in the test asset name in the **Search** field.
- Click the **Filter** icon  in the **SUITES** or **TESTS** tab to filter the displayed assets based on the asset type.

For example, select **API Suite** in the **SUITES** tab to display all API Suites or select **Functional Test** in the **TESTS** tab to display all functional tests that are in the selected branch of the repository.

- Click the **Filter** icon  in the **ADVANCED** tab, and then create a filter query by using the **New filter** option by performing the following steps:
 - Click **New filter**.
 - Enter a name for the filter.

- c. Select an operator, and add a rule or a group of rules.
- d. Add or enter the relevant parameters and either select or enter the condition and the criteria for the condition.

You can select a parameter from the following list:

- Type
 - Test Asset Name
 - Test Asset Path
 - Last Result
 - Next Run
 - Components
- e. Save the filter query to save and apply the filter query to filter the assets based on the query.

The test assets that match the filter criteria are displayed.

- Retrieve and apply a saved filter query, if you have saved filter queries previously by performing the following steps:



Note: The filter query applied previously is selected and the assets based on that filter query are displayed. To apply a different filter query, you must have created and saved the filter query.

- a. Click the **Filter** icon  in the **ADVANCED** tab.


The filter queries that you created and saved are displayed.

- b. Click the filter that you want to apply.

The test assets that match the filter criteria are displayed.

You have identified the test asset that you want to run.

Clicking the test name displays the **Details** panel. You can view the details of the test such as the description, the branch in the repository that contains the asset, the Git details, and the details of the commits to the repository. You can also view the history of the test runs for the specific test under the **History** tab of the **Details** panel.

6. Click the **Execute** icon  in the row of the identified test asset.

The **Execute test asset** dialog is displayed.



Notes:



- If you have configured some or all of the settings for the current test run, and you do not want to continue with those settings, you can reset the settings by clicking **Reset**.
- If you want to repeat a test run and do not want to use the saved settings from a previous run, you can reset all the saved settings to their default values by clicking **Reset**.

7. Select the version of the test resources that you want to run by completing any of the following actions:

- Expand the list in the **Version** field, find the version of the test resources, and then select the version.

Use the following details about the version of the test resources that are displayed to identify the version that you want:

- Commit message.
- Tags labeled by the user for the version committed.
- The user who committed the version to the repository.
- Relative time of the commit. For example, *2 hours ago* or *3 days ago*.

The list displays the versions of the test resources committed by all users to the branch in the repository. The versions are arranged with the latest version that is committed, and then followed by the versions committed previously.

- Expand the list in the **Version** field, and then search for the version that you want to select by entering a partial or the complete commit message of that version.

The version that matches the search criteria is displayed and it is selected for the test run.

8. Select the time for scheduling the test run from the following options:

- No action is required if you want to initiate the test run immediately after you click **Execute**.





Important: Click **Execute** only after you have configured the other settings in this dialog.




Note: The default time for scheduling a run is **Now**.

- Select **Schedule** and perform the following actions if you want to schedule a single test run or configure recurring test runs:

Schedule	Actions
A single test run	Perform the following steps: <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start.


Schedule	Actions
	<p>d. Select the Never option in the Repeat every field.</p> <p>e. Click Save.</p> <p>The test run is scheduled to run at the selected time on the scheduled date.</p>
<p>Recurring test runs</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Set the frequency at which the test runs must run by entering the number in the Repeat every field, and then select the period from the list. You can select from the following options: <ul style="list-style-type: none"> ▪ Minute ▪ Hour ▪ Day ▪ Week ▪ Month <p>For example, if you want the test run to be run every day at the set time, enter 1 and, then select the Day(s) option.</p> <p>You can also schedule the test to run at the set time on specific weekdays by selecting the Week(s) option. For example, the following image displays the days selected for the test runs as <i>Monday, Wednesday, and Friday</i>:</p> <div data-bbox="708 1283 1421 1444" style="border: 1px solid black; padding: 5px; margin: 10px 0;"> <p>Repeat every <input style="width: 60px;" type="text" value="1"/> <input style="width: 100px;" type="text" value="Week(s)"/> ▾</p> <p style="text-align: center;"> <input type="radio"/> S <input checked="" type="radio"/> M <input type="radio"/> T <input checked="" type="radio"/> W <input type="radio"/> T <input checked="" type="radio"/> F <input type="radio"/> S </p> </div> <p>e. Set the option to end the recurring test runs from the following options:</p>

Schedule	Actions
	<ul style="list-style-type: none"> ▪ Select the Never option, if you do not want the recurring test runs to end. ▪ Select the On option, and then click the Calendar icon . You can select the date and time after which the scheduled test runs do not run. <p>f. Click Save.</p> <p>The recurring test runs are scheduled to start the first run at the selected time on the scheduled date, and to repeat the run at the frequency that you set.</p>

9. Enter a label for the test run that helps you to identify the test on the **Results** page.

After the test run completes, the label that you entered is displayed for the test under the **Labels** column on the **Results** page. After you have created a label, any member of the project can use that label.

The default value for the **Label** field is null or an empty field.

 **Important:** The configuration that you set for the test run in the **Execute test asset** dialog is preserved when you run the same test again. Those changes are not visible when another user logs in to Rational® Test Automation Server. For example, if you created new variables on the server, those variables are available only for you when the same test is run again.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.


10. Click **Advanced** to make the following advanced configurations:

- a. Enter any JVM arguments that must be passed to the test run at run time in the **JVM Arguments** field, if applicable for the test.

You can enter a variable that points to the *MAVEN_OPTS* property.

- b. Enter the environment variables that must be passed to the test run at run time in the **Environment Variables** field, if applicable for the test.

For example, if you have created the JUnit tests by using JDK V12, then you must enter the variable as **JDKOTHER** and the value as **jdkv12**, so that the test run uses the JDK in the path `/data/junit-ext/jdks/jdkv12`.

 **Note:** You must separate the arguments or variables with a white space when you enter them in the same line or start each argument or variable on a new line.


The default value for each of the fields for the advanced settings is null or an empty field.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

11. Follow the instructions if you want to export the test results to a Jira issue in your Xray project in Jira:

a. Click the **RESULTS** tab.

b. Perform the actions listed in the following table to select an option to export the test results:


Export options	Results in...	Actions
Jira Xray	<p>The test results of the test that you configure for a run are exported to the selected issue after the test run is completed. The results are exported to the selected issue that exists in your Xray project in Jira.</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> i. Select the Jira Xray option. <p>The Select issue field is displayed.</p> <ol style="list-style-type: none"> ii. Click the field to view the list of issues in the Xray project in Jira. Alternatively, you can enter the issue key in the Jira Xray project. iii. Select the issue to which you want to export the test results. iv. Continue with the next step.
New Jira item	<p>The test results of the test that you configure for a run are exported to a new Jira issue. The issue is created in your project in Jira after the test run is completed.</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> i. Select the New Jira item option. ii. Continue with the next step.
None	<p>The test results are not exported.</p> <p> Note: This is the default option.</p>	<p>Perform the following steps when:</p> <ul style="list-style-type: none"> ▪ You do not want to export the test results. ▪ You selected a Jira issue in a previous run of the same test, and you do not want to export results of the current run. <ol style="list-style-type: none"> i. Select the None option. ii. Continue with the next step.

12. Follow the instructions if you want to change the location of a Kubernetes cluster for running the test:

a. Click the **LOCATION** tab, if it is not already open.

The **Default Cluster** is the default location where the test runs, and it is listed under the **Host** column.

The information about the availability of the default location is displayed.

 **Important:** You must have added the registered remote Kubernetes clusters to your project that are then displayed under the **Override** column:

 **Notes:**

- If the remote clusters are not added to your project, the option **No override options** is displayed as the default value and the test runs in the Kubernetes cluster of Rational® Test Automation Server.
- If remote clusters are added to your project, the added clusters are displayed along with their availability status and ownership information.

b. Select the location where you want to run the test asset from the following options:

- Select the **Default Cluster** when no remote clusters are available in your project.
- Select the remote cluster from the list when a remote cluster is available in your project.
- Select **No override options**, if you selected any remote cluster and want to revert to the **Default Cluster** to run the test asset.

c. Click **Execute**.


The test run is initiated.

Results

You have started or scheduled a test run of a JUnit test.

What to do next

You can choose to perform any of the following tasks:

- Stop the test run at any point after the test run is initiated from the **Execution** page. See [Stopping test runs on page 403](#).
- Cancel a scheduled test run from the **Execution** page. See [Canceling scheduled test runs on page 406](#).
- View all the states of the test asset by clicking the **Show in the Progress page** icon  for the test asset for which you started or scheduled the run. See [Viewing the state of test assets on page 394](#).
- View the progress of the test from the **Progress** page. See [Viewing the progress of running test assets on page 398](#).
- View the results, reports, and logs of the test from the **Results** page after the test completes the run. See [Test results on page 455](#).

Related information

[Resetting the configuration settings for a test run on page 402](#)

[Test run configurations on page 249](#)

Configuring a run of a Postman test

After you added the test resources that you created and exported from Postman to the server project, you can configure the Postman tests to be run on IBM® Rational® Test Automation Server or a remote Kubernetes cluster.

Before you begin

You must have completed the following tasks:

- Read and completed the tasks mentioned in [Test run considerations for Postman tests on page 208](#).
- Read and completed the tasks mentioned in [Considerations for using a remote Kubernetes cluster on page 235](#), if you want to use a remote Kubernetes cluster as a location to run the test.
- Completed the Xray integration, if you want to export the test results to a Jira issue in your Xray project in Jira. See [Integration with Xray test management for Jira on page 593](#).
- Ensured that the Postman extension in IBM® Rational® Test Automation Server is enabled before you configure a test run.
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created a project in your team space in Rational® Test Automation Server. See [Adding a project on page 619](#).
- Added the remote Git repository that contains the test resources to the project. See [Adding repositories to a server project on page 620](#).

About this task



Important: You can run the Postman tests only on the default cluster of IBM® Rational® Test Automation Server or a remote Kubernetes cluster. You cannot run the Postman tests on remote agents or remote Docker hosts that are registered with IBM® Rational® Test Automation Server.

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the test assets, and then click **Execution**.
3. Select the branch of the repository that contains the test assets.

The test assets that are contained in the selected branch of the repository are displayed in the following tabs on the **Execution** page:

Tab	Description
SUITES	Lists all suites, compound tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules that are in the selected branch of the repository.

Tab	Description
TESTS	Lists all API tests, functional tests, or performance tests that are in the selected branch of the repository.
ADVANCED	Lists all assets that are displayed when custom filters are applied for assets in the selected branch of the repository.

4. Select the tab based on the type of test assets that you want to run as indicated in the following table:


If...	Then...
You want to run AFT Suites, API Suites, Compound Tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules,.	Click the SUITES tab.
You want to run API tests, functional tests, or performance tests.	Click the TESTS tab.
You want to find Suites and tests by using custom filters, and then run Suites and tests.	Click the ADVANCED tab.










The test assets in the selected tab are displayed.


5. Identify the test asset that you want to run by performing any of the following actions:




























- Scroll through the list.


 **Tip:** You can hover over the icon in the **Type** column to know the type of the test asset.

 **Note:** You can also identify the type of the asset from the icon that represents the test type as shown in the following table:


Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the ADVANCED tab
	API test			
	Functional test			
	Performance test			



Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the ADVANCED tab
	AFT Suite			
	API Suite			
	Compound Test			
	HCL AppScan CodeSweep			
	JMeter Test			
	JUnit Test			
	Postman test			
	Rate Schedule			
	VU Schedule			

- Search for the test asset by entering any text contained in the test asset name in the **Search** field.
- Click the **Filter** icon  in the **SUITES** or **TESTS** tab to filter the displayed assets based on the asset type.

For example, select **API Suite** in the **SUITES** tab to display all API Suites or select **Functional Test** in the **TESTS** tab to display all functional tests that are in the selected branch of the repository.

- Click the **Filter** icon  in the **ADVANCED** tab, and then create a filter query by using the **New filter** option by performing the following steps:
 - Click **New filter**.
 - Enter a name for the filter.
 - Select an operator, and add a rule or a group of rules.
 - Add or enter the relevant parameters and either select or enter the condition and the criteria for the condition.

You can select a parameter from the following list:

- Type
- Test Asset Name

- Test Asset Path
 - Last Result
 - Next Run
 - Components
- e. Save the filter query to save and apply the filter query to filter the assets based on the query.

The test assets that match the filter criteria are displayed.

- Retrieve and apply a saved filter query, if you have saved filter queries previously by performing the following steps:



Note: The filter query applied previously is selected and the assets based on that filter query are displayed. To apply a different filter query, you must have created and saved the filter query.

- a. Click the **Filter** icon  in the **ADVANCED** tab.


The filter queries that you created and saved are displayed.

- b. Click the filter that you want to apply.

The test assets that match the filter criteria are displayed.

You have identified the test asset that you want to run.

Clicking the test name displays the **Details** panel. You can view the details of the test such as the description, the branch in the repository that contains the asset, the Git details, and the details of the commits to the repository. You can also view the history of the test runs for the specific test under the **History** tab of the **Details** panel.

6. Click the **Execute** icon  in the row of the identified test asset.

The **Execute test asset** dialog is displayed.



Notes:

- If you have configured some or all of the settings for the current test run, and you do not want to continue with those settings, you can reset the settings by clicking **Reset**.
- If you want to repeat a test run and do not want to use the saved settings from a previous run, you can reset all the saved settings to their default values by clicking **Reset**.

7. Select the version of the test resources that you want to run by completing any of the following actions:



Note: The test resources in the version can contain the test assets, datasets, AFT XML files, API environment tags, and other resources specific to projects created from any of the desktop clients.

- Expand the list in the **Version** field, find the version of the test resources, and then select the version.

Use the following details about the version of the test resources that are displayed to identify the version that you want:

- Commit message.
- Tags labeled by the user for the version committed.
- The user who committed the version to the repository.
- Relative time of the commit. For example, *2 hours ago* or *3 days ago*.

The list displays the versions of the test resources committed by all users to the branch in the repository. The versions are arranged with the latest version that is committed, and then followed by the versions committed previously.

- Expand the list in the **Version** field, and then search for the version that you want to select by entering a partial or the complete commit message of that version.

The version that matches the search criteria is displayed and it is selected for the test run.

The default value for the version selected for the run is the latest version in the selected branch of the repository. If you do not select any version, then the latest version is selected for the test run.



Notes:

- If you selected a version but you do not want to use that version in the test run, you can remove the selected version by clicking the ✕ icon. As a result, the default version is selected for the test run.
- If you repeated a test or ran the test again from the **Results** page, then the version of the test resources that you chose for the earlier run is shown as selected. You can either retain this version or select any other version from the list. You can also remove the previous version by clicking the ✕ icon.

8. Select the time for scheduling the test run from the following options:

- No action is required if you want to initiate the test run immediately after you click **Execute**.



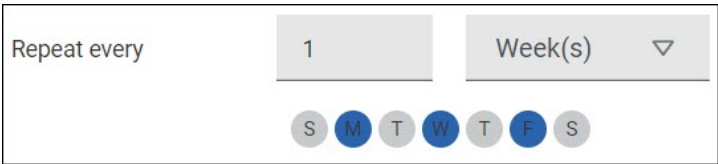



Important: Click **Execute** only after you have configured the other settings in this dialog.



Note: The default time for scheduling a run is **Now**.

- Select **Schedule** and perform the following actions if you want to schedule a single test run or configure recurring test runs:


Schedule	Actions
A single test run	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Select the Never option in the Repeat every field. Click Save. <p>The test run is scheduled to run at the selected time on the scheduled date.</p>
Recurring test runs	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Set the frequency at which the test runs must run by entering the number in the Repeat every field, and then select the period from the list. You can select from the following options: <ul style="list-style-type: none"> ▪ Minute ▪ Hour ▪ Day ▪ Week ▪ Month <p>For example, if you want the test run to be run every day at the set time, enter 1 and, then select the Day(s) option.</p> <p>You can also schedule the test to run at the set time on specific weekdays by selecting the Week(s) option. For example, the following image displays the days selected for the test runs as <i>Monday, Wednesday, and Friday</i>:</p> <div data-bbox="708 1482 1421 1644" data-label="Image">  <p>The image shows a user interface for scheduling recurring test runs. It features a 'Repeat every' field with a dropdown menu. The field contains the number '1' and the text 'Week(s)' with a downward arrow. Below this field is a row of seven circular buttons representing the days of the week: S (Sunday), M (Monday), T (Tuesday), W (Wednesday), T (Thursday), F (Friday), and S (Saturday). The M, W, and F buttons are highlighted in blue, indicating they are selected for the test runs.</p> </div> Set the option to end the recurring test runs from the following options:

Schedule	Actions
	<ul style="list-style-type: none"> ▪ Select the Never option, if you do not want the recurring test runs to end. ▪ Select the On option, and then click the Calendar icon . You can select the date and time after which the scheduled test runs do not run. <p>f. Click Save.</p> <p>The recurring test runs are scheduled to start the first run at the selected time on the scheduled date, and to repeat the run at the frequency that you set.</p>

9. Enter a label for the test run that helps you to identify the test on the **Results** page.

After the test run completes, the label that you entered is displayed for the test under the **Labels** column on the **Results** page. After you have created a label, any member of the project can use that label.

The default value for the **Label** field is null or an empty field.


 **Important:** The configuration that you set for the test run in the **Execute test asset** dialog is preserved when you run the same test again. Those changes are not visible when another user logs in to Rational® Test Automation Server. For example, if you created new variables on the server, those variables are available only for you when the same test is run again.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

10. Click **Advanced**, and then enter program arguments that must be passed to the test run at run time in the **Program Arguments**, if applicable for the test.

You can enter the command options that you use when you run Postman tests by using Newman, as the program arguments.

For example, enter **--verbose**, if you want the details of the test run and each request sent.


 **Note:** You must separate the arguments or variables with a white space when you enter them in the same line or start each argument or variable on a new line.

The default value for each of the fields for the advanced settings is null or an empty field.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

11. Follow the instructions if you want to export the test results to a Jira issue in your Xray project in Jira:


- a. Click the **RESULTS** tab.
- b. Perform the actions listed in the following table to select an option to export the test results:

Export options	Results in...	Actions
Jira Xray	The test results of the test that you configure for a run are exported to the selected issue after the test run is completed. The results are exported to the selected issue that exists in your Xray project in Jira.	Perform the following steps: <ol style="list-style-type: none"> i. Select the Jira Xray option. <p>The Select issue field is displayed.</p> <ol style="list-style-type: none"> ii. Click the field to view the list of issues in the Xray project in Jira. Alternatively, you can enter the issue key in the Jira Xray project. iii. Select the issue to which you want to export the test results. iv. Continue with the next step.
New Jira item	The test results of the test that you configure for a run are exported to a new Jira issue. The issue is created in your project in Jira after the test run is completed.	Perform the following steps: <ol style="list-style-type: none"> i. Select the New Jira item option. ii. Continue with the next step.
None	The test results are not exported.  Note: This is the default option.	Perform the following steps when: <ul style="list-style-type: none"> ▪ You do not want to export the test results. ▪ You selected a Jira issue in a previous run of the same test, and you do not want to export results of the current run. <ol style="list-style-type: none"> i. Select the None option. ii. Continue with the next step.

12. Follow the instructions if you want to change the location of a Kubernetes cluster for running the test:

- a. Click the **LOCATION** tab, if it is not already open.

The **Default Cluster** is the default location where the test runs, and it is listed under the **Host** column. The information about the availability of the default location is displayed.

 **Important:** You must have added the registered remote Kubernetes clusters to your project that are then displayed under the **Override** column:



Notes:

- If the remote clusters are not added to your project, the option **No override options** is displayed as the default value and the test runs in the Kubernetes cluster of Rational® Test Automation Server.
- If remote clusters are added to your project, the added clusters are displayed along with their availability status and ownership information.

b. Select the location where you want to run the test asset from the following options:

- Select the **Default Cluster** when no remote clusters are available in your project.
- Select the remote cluster from the list when a remote cluster is available in your project.
- Select **No override options**, if you selected any remote cluster and want to revert to the **Default Cluster** to run the test asset.

c. Click **Execute**.


The test run is initiated.

Results

You have started or scheduled a test run of a Postman test.

What to do next

You can choose to perform any of the following tasks:

- Stop the test run at any point after the test run is initiated from the **Execution** page. See [Stopping test runs on page 403](#).
- Cancel a scheduled test run from the **Execution** page. See [Canceling scheduled test runs on page 406](#).
- View all the states of the test asset by clicking the **Show in the Progress page** icon  for the test asset for which you started or scheduled the run. See [Viewing the state of test assets on page 394](#).
- View the progress of the test from the **Progress** page. See [Viewing the progress of running test assets on page 398](#).
- View the results, reports, and logs of the test from the **Results** page after the test completes the run. See [Test results on page 455](#).

Related information

[Resetting the configuration settings for a test run on page 402](#)

[Test run configurations on page 249](#)

Configuring a run of a Rate Schedule or VU Schedule

After you added the test resources that you created in the desktop client to the project, you can configure a Rate Schedule or VU Schedule to be run on IBM® Rational® Test Automation Server or a remote Kubernetes cluster.

Before you begin

You must have completed the following tasks:

- Created Schedules in Rational® Performance Tester and added the test asset to the project repository on IBM® Rational® Test Automation Server.
- Read and completed the tasks mentioned in [Prerequisites to running tests on page 198](#), if they apply to the test that you want to configure for a run.
- Read [Considerations for using Jaeger traces in reports on page 205](#), if you want to configure a run and you want Jaeger to report the test results for the test.
- Read [Test run considerations for schedules on page 203](#), if you want to configure a run for Rate Schedules or VU Schedules that have Resource Monitoring sources configured.
- Read and completed the tasks mentioned in [Test run considerations for running tests on remote performance agents on page 211](#), if you want to use a remote agent as the location to run the test.
- Read and completed the tasks mentioned in [Considerations for using a remote Kubernetes cluster on page 235](#), if you want to use a remote Kubernetes cluster as a location to run the test.
- Completed the Xray integration, if you want to export the test results to a Jira issue in your Xray project in Jira. See [Integration with Xray test management for Jira on page 593](#).
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created a project in your team space in Rational® Test Automation Server. See [Adding a project on page 619](#).
- Added the remote Git repository that contains the test resources to the project. See [Adding repositories to a server project on page 620](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the test assets, and then click **Execution**.
3. Select the branch of the repository that contains the test assets.

The test assets that are contained in the selected branch of the repository are displayed in the following tabs on the **Execution** page:


Tab	Description
SUITES	Lists all suites, compound tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules that are in the selected branch of the repository.


































Tab	Description
TESTS	Lists all API tests, functional tests, or performance tests that are in the selected branch of the repository.
ADVANCED	Lists all assets that are displayed when custom filters are applied for assets in the selected branch of the repository.

4. Identify the test asset that you want to run by performing any of the following actions:




- Scroll through the list.


 **Tip:** You can hover over the icon in the **Type** column to know the type of the test asset.

 **Note:** You can also identify the type of the asset from the icon that represents the test type as shown in the following table:


Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the AD-VANCED tab
	API test			
	Functional test			
	Performance test			
	AFT Suite			
	API Suite			
	Compound Test			
	HCL AppScan CodeSweep			
	JMeter Test			
	JUnit Test			
	Postman test			
	Rate Schedule			



Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the AD-VANCED tab
	VU Schedule			

- Search for the test asset by entering any text contained in the test asset name in the **Search** field.
- Click the **Filter** icon  in the **SUITES** or **TESTS** tab to filter the displayed assets based on the asset type.

For example, select **API Suite** in the **SUITES** tab to display all API Suites or select **Functional Test** in the **TESTS** tab to display all functional tests that are in the selected branch of the repository.

- Click the **Filter** icon  in the **ADVANCED** tab, and then create a filter query by using the **New filter** option by performing the following steps:
 - Click **New filter**.
 - Enter a name for the filter.
 - Select an operator, and add a rule or a group of rules.
 - Add or enter the relevant parameters and either select or enter the condition and the criteria for the condition.

You can select a parameter from the following list:

- Type
 - Test Asset Name
 - Test Asset Path
 - Last Result
 - Next Run
 - Components
- Save the filter query to save and apply the filter query to filter the assets based on the query.

The test assets that match the filter criteria are displayed.

- Retrieve and apply a saved filter query, if you have saved filter queries previously by performing the following steps:



Note: The filter query applied previously is selected and the assets based on that filter query are displayed. To apply a different filter query, you must have created and saved the filter query.

- a. Click the **Filter** icon  in the **ADVANCED** tab.


The filter queries that you created and saved are displayed.

- b. Click the filter that you want to apply.

The test assets that match the filter criteria are displayed.

You have identified the test asset that you want to run.

Clicking the test name displays the **Details** panel. You can view the details of the test such as the description, the branch in the repository that contains the asset, the Git details, and the details of the commits to the repository. You can also view the history of the test runs for the specific test under the **History** tab of the **Details** panel.

5. Click the **Execute** icon  in the row of the identified test asset.

The **Execute test asset** dialog is displayed.



Notes:

- If you have configured some or all of the settings for the current test run, and you do not want to continue with those settings, you can reset the settings by clicking **Reset**.
- If you want to repeat a test run and do not want to use the saved settings from a previous run, you can reset all the saved settings to their default values by clicking **Reset**.

6. Select the version of the test resources that you want to run by completing any of the following actions:



Note: The test resources in the version can contain the test assets, datasets, AFT XML files, API environment tags, and other resources specific to projects created from any of the desktop clients.

- Expand the list in the **Version** field, find the version of the test resources, and then select the version.

Use the following details about the version of the test resources that are displayed to identify the version that you want:

- Commit message.
- Tags labeled by the user for the version committed.
- The user who committed the version to the repository.
- Relative time of the commit. For example, *2 hours ago* or *3 days ago*.

The list displays the versions of the test resources committed by all users to the branch in the repository. The versions are arranged with the latest version that is committed, and then followed by the versions committed previously.

- Expand the list in the **Version** field, and then search for the version that you want to select by entering a partial or the complete commit message of that version.

The version that matches the search criteria is displayed and it is selected for the test run.

The default value for the version selected for the run is the latest version in the selected branch of the repository. If you do not select any version, then the latest version is selected for the test run.



Notes:

- If you selected a version but you do not want to use that version in the test run, you can remove the selected version by clicking the ✕ icon. As a result, the default version is selected for the test run.
- If you repeated a test or ran the test again from the **Results** page, then the version of the test resources that you chose for the earlier run is shown as selected. You can either retain this version or select any other version from the list. You can also remove the previous version by clicking the ✕ icon.

7. Select the time for scheduling the test run from the following options:

- No action is required if you want to initiate the test run immediately after you click **Execute**.






Important: Click **Execute** only after you have configured the other settings in this dialog.



Note: The default time for scheduling a run is **Now**.

- Select **Schedule** and perform the following actions if you want to schedule a single test run or configure recurring test runs:


Schedule	Actions
A single test run	Perform the following steps: <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Select the Never option in the Repeat every field. Click Save.

Schedule	Actions
	<p>The test run is scheduled to run at the selected time on the scheduled date.</p>
<p>Recurring test runs</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Set the frequency at which the test runs must run by entering the number in the Repeat every field, and then select the period from the list. You can select from the following options: <ul style="list-style-type: none"> Minute Hour Day Week Month <p>For example, if you want the test run to be run every day at the set time, enter 1 and, then select the Day(s) option.</p> <p>You can also schedule the test to run at the set time on specific weekdays by selecting the Week(s) option. For example, the following image displays the days selected for the test runs as <i>Monday, Wednesday, and Friday</i>:</p> <div data-bbox="708 1178 1421 1337" data-label="Image"> </div> Set the option to end the recurring test runs from the following options: <ul style="list-style-type: none"> Select the Never option, if you do not want the recurring test runs to end. Select the On option, and then click the Calendar icon . You can select the date and time after which the scheduled test runs do not run. Click Save. <p>The recurring test runs are scheduled to start the first run at the selected time on the scheduled date, and to repeat the run at the frequency that you set.</p>

8. Enter a label for the test run that helps you to identify the test on the **Results** page.

After the test run completes, the label that you entered is displayed for the test under the **Labels** column on the **Results** page. After you have created a label, any member of the project can use that label.

The default value for the **Label** field is null or an empty field.

 **Important:** The configuration that you set for the test run in the **Execute test asset** dialog is preserved when you run the same test again. Those changes are not visible when another user logs in to Rational® Test Automation Server. For example, if you created new variables on the server, those variables are available only for you when the same test is run again.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

9. Click **Advanced** to make the following advanced configurations:

- a. Enter any JVM arguments that must be passed to the test run at run time in the **JVM Arguments** field, if applicable for the test.

For example, you can set a maximum Java heap size.

- b. Enter program arguments that must be passed to the test run at run time in the **Program Arguments**, if applicable for the test.

If the test that you want to configure supports the presentation of the test results data in different formats, then do the required tasks in the following scenarios:

If...	Then...
You want the test results data in different supported formats.	Enter <code>-history cisterna</code> in the Program Arguments field.
You want to obtain the test results as a Jaeger trace.	Enter <code>-history jaeger</code> in the Program Arguments field.
You want the report as a test log and as a Jaeger trace.	Enter <code>-history jaeger, testlog</code> in the Program Arguments field.
Jaeger is already set as a program argument and you want to remove the Jaeger trace for your test.	Delete the <code>-history jaeger</code> entry in the Program Arguments field.



Note: The default report format is the *test log* format for the test reports.

- c. Enter the environment variables that must be passed to the test run at run time in the **Environment Variables** field, if applicable for the test.

For example, enter the environment variables when the third-party libraries that are used in the test run refer to the environment variables for configuration.



Note: You must separate the arguments or variables with a white space when you enter them in the same line or start each argument or variable on a new line.

The default value for each of the fields for the advanced settings is null or an empty field.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

10. Follow the instructions if you are running a test asset that contains datasets:

- a. Click the **DATA SOURCES** tab, if it is not already open.
- b. Consider the following information about datasets before you select a dataset:

The default value for the datasets in the **DATA SOURCES** tab is null if the test asset did not have an associated dataset. If the asset had an associated dataset, the default value is the associated dataset.

You can utilize the dataset stored as an Excel or CSV file to override the original dataset associated with the Suite, test, or schedule. For example, when you have associated a dataset in `.xlsx`, `.xls`, or `.csv` format with the test or schedule in desktop clients and if you have another set of data stored in an Excel or CSV file, then you can select that dataset from the **Override** list.



Remember: You must have uploaded the dataset as an Excel or CSV file into the Git repository, and ensured that both the original dataset (from the test asset) and new datasets (added to the project) have the same column names.

- c. Select the dataset that you want to use in the test run from any of the following options:
 - Select the dataset that is displayed as the default dataset when the test asset contains a single dataset.



Note: If there is only one dataset in the test asset, then that dataset is displayed as the default dataset.

- Select the dataset from the list.



Note: If there are multiple datasets in the test asset, the datasets are listed in their increasing alphabetical order.

- Select the dataset from the **Override** list to override the dataset that was associated with the test in the desktop client.





Important: If the test contains an encrypted dataset, the Project Owner must classify it in the **DATA SECURITY** tab on the Project page before you can select it. You must have added datasets to your project from the **Dataset** page for the datasets to be displayed in the **Override** list.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

11. Follow the instructions if the test requires a variable that must be passed to the test at the test run time.

a. Click the **VARIABLES** tab, if it is not already open.

b. Choose one of the following methods to add the variables:

- To add new variables manually, click the **Add Variable** icon , enter the name, and value of the variable.
- To add new variables from your local computer or from the Git repository that is associated with your server project, click the **Upload** icon  and select the **Upload from local system** or **Browse from server** to select the variable file.



Note: You must have created a file with the variables before you can select the file.

The default value for the variables is null or an empty field.

If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.


12. Follow the instructions if you want to override the Resource Monitoring labels in the test asset with the Resource Monitoring labels that you created in Rational® Test Automation Server:

a. Click the **Resource Monitoring** tab.



Note:

The **Resource Monitoring** tab displays only if you enabled the **Resource Monitoring from Service** option in Rational® Performance Tester when you created the test asset.

b. Click , and press the `Ctrl + Space bar` keys, or enter the initial letter of a label to select a label in the list.




Note:

You can select or add labels only if you added the labels in the **Resource Monitoring Sources** page in Rational® Test Automation Server.

13. Follow the instructions if you want to export the test results to a Jira issue in your Xray project in Jira:

- a. Click the **RESULTS** tab.
- b. Perform the actions listed in the following table to select an option to export the test results:

Export options	Results in...	Actions
Jira Xray	<p>The test results of the test that you configure for a run are exported to the selected issue after the test run is completed. The results are exported to the selected issue that exists in your Xray project in Jira.</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> i. Select the Jira Xray option. <p>The Select issue field is displayed.</p> <ol style="list-style-type: none"> ii. Click the field to view the list of issues in the Xray project in Jira. Alternatively, you can enter the issue key in the Jira Xray project. iii. Select the issue to which you want to export the test results. iv. Continue with the next step.
New Jira item	<p>The test results of the test that you configure for a run are exported to a new Jira issue. The issue is created in your project in Jira after the test run is completed.</p>	<p>Perform the following steps:</p> <ol style="list-style-type: none"> i. Select the New Jira item option. ii. Continue with the next step.
None	<p>The test results are not exported.</p> <p> Note: This is the default option.</p>	<p>Perform the following steps when:</p> <ul style="list-style-type: none"> ▪ You do not want to export the test results. ▪ You selected a Jira issue in a previous run of the same test, and you do not want to export results of the current run. <ol style="list-style-type: none"> i. Select the None option. ii. Continue with the next step.

14. Select any of the following steps that depends on the location to run the test:

- If you want to run the tests on the remote agents, go to [Step 15 on page 381](#).
 - If you want to run the tests on a Kubernetes cluster, go to [Step 16 on page 382](#).
15. Follow the instructions if you want to run a test on the remote performance agent that is connected either to the default Kubernetes cluster or the remote Kubernetes cluster:

- a. Click the **LOCATION** tab, if it is not already open.

The performance agents that are configured in the test asset are listed under the **Host** column. The information about the availability of the performance agent is displayed.



Note: You must have added performance agents to your project from the **Agents and Intercepts** page for the performance agents to be displayed under the **Override** column.

The default value for the performance agents is null or an empty field if no performance agents were configured in the test asset. If the test asset contains performance agents that are configured, then the default performance agent is the first item to be displayed on the list of performance agents listed in the increasing alphabetical order.

- b. Select the performance agent where you want to run the test asset in the following scenarios:

If...	Then...	Action
The performance agent that is specified in the test assets is available and no other performance agents are added to the project.	The performance agent that is specified in the test assets is displayed in the Override column.	Select the performance agent in the Host that is also displayed in the Override column as the location to run the test.
The performance agent that is specified in the test assets is not available and no other performance agents are added to the project.	There is no performance agent displayed for override in the Override column.	You cannot run the test on the performance agent. You must wait until the performance agent is available to run the test on the agent.
The agent that is specified in the test assets is available and there are other agents that are added to the project.	Performance agents that have capabilities that match with the agent capabilities specified in the test assets are listed in the Override column as follows: <ul style="list-style-type: none"> ▪ Performance agents with capabilities that best match the capabil- 	Perform any of the following actions: <ul style="list-style-type: none"> ▪ Select the performance agent in the Host that is also displayed in the Override column as the location to run the test.

If...	Then...	Action
	<p>ities in the agent in the asset are at the top of the list.</p> <ul style="list-style-type: none"> ▪ Performance agents with capabilities that do not match with the capabilities in the agent in the asset are listed subsequently. <p>You can find details of both performance agents for the matching capabilities when you hover over the performance agent in the Override column.</p>	<ul style="list-style-type: none"> ▪ Select a performance agent in the Override column as the location to run the test.


c. Click **Execute**.

The test run is initiated.

16. Follow the instructions if you want to change the location of a Kubernetes cluster for running the test:

a. Click the **LOCATION** tab, if it is not already open.

The **Default Cluster** is the default location where the test runs, and it is listed under the **Host** column. The information about the availability of the default location is displayed.

 **Important:** You must have added the following remote hosts to your project that are then displayed under the **Override** column:

- Docker hosts that are registered with Rational® Test Automation Server.
- Kubernetes clusters that are registered with a team space.

 **Notes:**

- If the remote Docker hosts or clusters are not added to your project, the option **No override options** is displayed as the default value and the test runs in the Kubernetes cluster of Rational® Test Automation Server.
- If remote Docker hosts or clusters are added to your project, the added Docker hosts or clusters are displayed along with their availability status and ownership information.

b. Select the location where you want to run the test asset from the following options:

- Select the **Default Cluster** when no remote Docker hosts or clusters are available in your project.
- Select the remote Docker host or cluster from the list when a remote Docker host or cluster is available in your project.
- Select **No override options**, if you selected any remote Docker host or cluster and want to revert to the **Default Cluster** to run the test asset.

c. Click **Execute**.


The test run is initiated.

Results

You have started or scheduled a test run of a Rate Schedule or VU Schedule.

What to do next

You can choose to perform any of the following tasks:

- Stop the test run at any point after the test run is initiated from the **Execution** page. See [Stopping test runs on page 403](#).
- Cancel a scheduled test run from the **Execution** page. See [Canceling scheduled test runs on page 406](#).
- View all the states of the test asset by clicking the **Show in the Progress page** icon  for the test asset for which you started or scheduled the run. See [Viewing the state of test assets on page 394](#).
- View the progress of the test from the **Progress** page. See [Viewing the progress of running test assets on page 398](#).
- View the results, reports, and logs of the test from the **Results** page after the test completes the run. See [Test results on page 455](#).

Related information

[Resetting the configuration settings for a test run on page 402](#)

[Test run configurations on page 249](#)

Configuration of a scan of an application code

You can use Rational® Test Automation Server to scan your application code for security vulnerabilities similar to using the HCL AppScan CodeSweep plug-in to scan code in Visual Studio.

Rational® Test Automation Server is enabled to support scanning of your application code that is in a remote repository.

Before you commit your application code to the remote repository, you must complete certain tasks. See [Considerations to scan an application code on page 384](#).

After you commit the assets to the remote repository, you must complete the following tasks:

1. Log in to Rational® Test Automation Server.
2. Open the project in your team space.
3. Add the remote repository to your project.
4. Open the **Execution page**, and then run a scan of the asset.

For detailed instructions, see [Configuring a run to scan the application code on page 386](#).

Considerations to scan an application code

When you want to use IBM® Rational® Test Automation Server to run a scan of the application code for security vulnerabilities, you must read the considerations that you must take into account and complete certain tasks.

Before you can run a scan of the application code on Rational® Test Automation Server, you must have completed the following tasks:

- Created a file that has the following characteristics:
 - The format of the file must be `yaml`.
 - The name must be `.codesweep`.
- Committed the following assets or resources to the remote repository:
 - The `.codesweep` file.
 - The application code can either be in the root directory or another folder in the root directory.



Note: The default root directory is the directory that contains the `.codesweep` file.

You can specify the following information in the `.codesweep` file:

- The name of the asset that must be displayed as the test asset in the **Execution** page of Rational® Test Automation Server. The name must be specified at the first level in the file as the value of the **name** attribute.
- Optionally, the name of folders that contain the application code.

All files and folders in the root directory are scanned as the default action, if no files or folders are specified under the **folders** attribute.

You can specify the files or folders to scan by using any of the following options:

- Specify the files or folders by using an `XML` file that is in the same format as the HCL AppScan IRX file.
- Specify the files or folders in the `.codesweep` file under the **includes** or **excludes** attributes under the **folders** attribute.

- Optionally, you can specify the name of files or folders that must be included in the scan. You must specify the files or folders as the value of the **includes** attribute under the **folders** attribute.
- Optionally, you can specify the name of files or folders that contain the application code that need not be scanned. You must specify the files or folders as the value of the **excludes** attribute under the **folders** attribute.



Note: You cannot use *wildcard* characters to indicate the files or folders anywhere in the `.codesweep` file.

- Optionally, you can specify the rules that are to be ignored in the scan. You can specify the rules by using any of the following options:
 - Specify a file, which is a copy of `~/appscan/custom_scanners_service_ignore.json` that is created by the CodeSweep plug-in for Visual Studio. With this method, you can reuse the preferences that you have configured in your development environment.
 - Specify a list of rules. Each rule is defined by a name and a language identifier.
- Optionally, you can specify the criteria for attributes and their values that specify the verdict of the scan and the range for the security errors in the code.

For example, you can consider the following details:

Where...	Is...
The name of the asset that is to be displayed on the Execution page	<code>mycode</code>
The folder that contains the code to be scanned	<code>use_code</code>
The folder that contains the code that is to be excluded in the scan	<code>do_not_use_code</code>
The value of the attribute <i>Verdict</i>	
The range for the security errors is classified as follows: <ul style="list-style-type: none"> • <code>max-high</code> • <code>max-medium</code> • <code>max-low</code> 	The conditions for a <i>pass</i> verdict are as follows: <ul style="list-style-type: none"> • <code>0</code> • <code>10</code> • <code>100</code>

The contents of the `.codesweep` file is as follows:

```
name: mycode
folders: # default=./, Optional
  configuration: configuration.xml # XML Configuration file, Optional
  includes: # list of folders to include, Optional
    - use_code
  excludes: # list of folders to exclude, Optional
```

```

- do_not_use_code
ignore: # Ignore rules, Optional
  file: custom_scanners_service_ignore.json # json file coming from visual code, Optional
  rules: #list of rules to ignore, Optional
    - name: com.ouncelabs.languageelite.nodejs.Rules.NodeJS_PrivateDataExposedviaHTTTPURL # Name,
      Required
      language: NodeJS # Language, Required
    - name: com.ouncelabs.languageelite.nodejs.Rules.NodeJS_PrivateDataExposedviaHTTP # Name,
      Required
      language: NodeJS # Language, Required
verdict: # PASS verdict, Optional
  max-high: 0 # default=0, Optional
  max-medium: 10 # default=0, Optional
  max-low: 100 # default=-1 (no limit), Optional

```

You can continue to configure a run of a scan of the code, see [Configuring a run to scan the application code on page 386](#).



Restriction: You cannot run the test assets on a remote Docker host.

Related information

[Configuration of a scan of an application code on page 383](#)

Configuring a run to scan the application code

After you added the assets in the remote repository to a project in your team space, you can configure a run to scan the application code specified in the `codesweep` file by using the execution service on IBM® Rational® Test Automation Server.

Before you begin

You must have completed the following tasks:

- Read and completed the tasks mentioned in [Considerations to scan an application code on page 384](#).
- Read and completed the tasks mentioned in [Considerations for using a remote Kubernetes cluster on page 235](#), if you want to use a remote Kubernetes cluster as a location to run the test.
- Completed the Xray integration, if you want to export the test results to a Jira issue in your Xray project in Jira. See [Integration with Xray test management for Jira on page 593](#).
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created a project in your team space in Rational® Test Automation Server. See [Adding a project on page 619](#).
- Added the remote Git repository that contains the test resources to the project. See [Adding repositories to a server project on page 620](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the test assets, and then click **Execution**.
3. Select the branch of the repository that contains the test assets.


The test assets that are contained in the selected branch of the repository are displayed in the following tabs on the **Execution** page:
















Tab	Description
SUITES	Lists all suites, compound tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules that are in the selected branch of the repository.
TESTS	Lists all API tests, functional tests, or performance tests that are in the selected branch of the repository.
ADVANCED	Lists all assets that are displayed when custom filters are applied for assets in the selected branch of the repository.


4. Identify the test asset or resource that you want to run by performing any of the following actions:






















- Scroll through the list.


 **Tip:** You can hover over the icon in the **Type** column to know the type of the test asset.

 **Note:** You can also identify the type of the asset from the icon that represents the test type as shown in the following table:


Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the ADVANCED tab
	API test			
	Functional test			
	Performance test			
	AFT Suite			
	API Suite			



Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the ADVANCED tab
	Compound Test			
	HCL AppScan CodeSweep			
	JMeter Test			
	JUnit Test			
	Postman test			
	Rate Schedule			
	VU Schedule			

- Search for the test asset or resource by entering any text contained in the test asset or resource name in the **Search** field.
- Click the **Filter** icon  in the **SUITES** or **TESTS** tab to filter the displayed assets based on the asset type.

For example, select **API Suite** in the **SUITES** tab to display all API Suites or select **Functional Test** in the **TESTS** tab to display all functional tests in the selected branch of the repository.

- Click the **Filter** icon  in the **ADVANCED** tab, and then create a filter query by using the **New filter** option by performing the following steps:
 - Click **New filter**.
 - Enter a name for the filter.
 - Select an operator, and add a rule, or a group of rules.
 - Add or enter the relevant parameters and either select or enter the condition and the criteria for the condition.

You can select a parameter from the following list:

- Type
- Test Asset Name
- Test Asset Path
- Last Result

- Next Run
 - Components
- e. Save the filter query to save and apply the filter query to filter the assets based on the query.

The test assets that match the filter criteria are displayed.

- Retrieve and apply a saved filter query, if you have saved filter queries previously by performing the following steps:




Note: The filter query applied previously is selected and the assets based on that filter query are displayed. To apply a different filter query, you must have created and saved the filter query.

- a. Click the **Filter** icon  in the **ADVANCED** tab.

The filter queries that you created and saved are displayed.

- b. Click the filter that you want to apply.

The test assets that match the filter criteria are displayed.

5. Click the **Execute** icon  in the row of the identified test asset.

The **Execute test asset** dialog is displayed.



Notes:

- If you have configured some or all of the settings for the current test run, and you do not want to continue with those settings, you can reset the settings by clicking **Reset**.
- If you want to repeat a test run and do not want to use the saved settings from a previous run, you can reset all the saved settings to their default values by clicking **Reset**.

6. Select the version of the test resources that you want to run by completing any of the following actions:

- Expand the list in the **Version** field, find the version of the test resources, and then select the version.

Use the following details about the version of the test resources that are displayed to identify the version that you want:

- Commit message.
- Tags labeled by the user for the version committed.
- The user who committed the version to the repository.
- Relative time of the commit. For example, *2 hours ago* or *3 days ago*.

The list displays the versions of the test resources committed by all users to the branch in the repository. The versions are arranged with the latest version that is committed, and then followed by the versions committed previously.

- Expand the list in the **Version** field, and then search for the version that you want to select by entering a partial or the complete commit message of that version.

The version that matches the search criteria is displayed and it is selected for the test run.

The default value for the version selected for the run is the latest version in the selected branch of the repository. If you do not select any version, then the latest version is selected for the test run.



Notes:

- If you selected a version but you do not want to use that version in the test run, you can remove the selected version by clicking the **✕** icon. As a result, the default version is selected for the test run.
- If you repeated a test or ran the test again from the **Results** page, then the version of the test resources that you chose for the earlier run is shown as selected. You can either retain this version or select any other version from the list. You can also remove the previous version by clicking the **✕** icon.

7. Select the time for scheduling the test run from the following options:

- No action is required if you want to initiate the test run immediately after you click **Execute**.






Important: Click **Execute** only after you have configured the other settings in this dialog.



Note: The default time for scheduling a run is **Now**.

- Select **Schedule** and perform the following actions if you want to schedule a single test run or configure recurring test runs:


Schedule	Actions
A single test run	Perform the following steps: <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Select the Never option in the Repeat every field. Click Save.

Schedule	Actions
	The test run is scheduled to run at the selected time on the scheduled date.
Recurring test runs	<p>Perform the following steps:</p> <ol style="list-style-type: none"> Click the Calendar icon  in the row of the Start field. Select the date. Select the time at which the test run must start. Set the frequency at which the test runs must run by entering the number in the Repeat every field, and then select the period from the list. You can select from the following options: <ul style="list-style-type: none"> Minute Hour Day Week Month <p>For example, if you want the test run to be run every day at the set time, enter 1 and, then select the Day(s) option.</p> <p>You can also schedule the test to run at the set time on specific weekdays by selecting the Week(s) option. For example, the following image displays the days selected for the test runs as <i>Monday, Wednesday, and Friday</i>:</p> <div data-bbox="708 1178 1421 1339" data-label="Form"> </div> <ol style="list-style-type: none"> Set the option to end the recurring test runs from the following options: <ul style="list-style-type: none"> Select the Never option, if you do not want the recurring test runs to end. Select the On option, and then click the Calendar icon . You can select the date and time after which the scheduled test runs do not run. Click Save. <p>The recurring test runs are scheduled to start the first run at the selected time on the scheduled date, and to repeat the run at the frequency that you set.</p>

8. Enter a label for the test run that helps you to identify the test on the **Results** page.

After the test run completes, the label that you entered is displayed for the test under the **Labels** column on the **Results** page. After you have created a label, any member of the project can use that label.

The default value for the **Label** field is null or an empty field.

 **Important:** The configuration that you set for the test run in the **Execute test asset** dialog is preserved when you run the same test again. Those changes are not visible when another user logs in to Rational® Test Automation Server. For example, if you created new variables on the server, those variables are available only for you when the same test is run again.


If you want to run the test immediately or at the scheduled time, click **Execute**, or continue with the next step.

9. Follow the instructions if you want to export the test results to a Jira issue in your Xray project in Jira:

a. Click the **RESULTS** tab.

b. Perform the actions listed in the following table to select an option to export the test results:


Export options	Results in...	Actions
Jira Xray	The test results of the test that you configure for a run are exported to the selected issue after the test run is completed. The results are exported to the selected issue that exists in your Xray project in Jira.	Perform the following steps: i. Select the Jira Xray option. The Select issue field is displayed. ii. Click the field to view the list of issues in the Xray project in Jira. Alternatively, you can enter the issue key in the Jira Xray project. iii. Select the issue to which you want to export the test results. iv. Continue with the next step.
New Jira item	The test results of the test that you configure for a run are exported to a new Jira issue. The issue is created in your project in Jira after the test run is completed.	Perform the following steps: i. Select the New Jira item option. ii. Continue with the next step.

Export options	Results in...	Actions
None	<p>The test results are not exported.</p> <p> Note: This is the default option.</p>	<p>Perform the following steps when:</p> <ul style="list-style-type: none"> ▪ You do not want to export the test results. ▪ You selected a Jira issue in a previous run of the same test, and you do not want to export results of the current run. <p>i. Select the None option.</p> <p>ii. Continue with the next step.</p>

10. Follow the instructions if you want to change the location of a Kubernetes cluster for running the test:

- a. Click the **LOCATION** tab, if it is not already open.

The **Default Cluster** is the default location where the test runs, and it is listed under the **Host** column. The information about the availability of the default location is displayed.

 **Important:** You must have added the registered remote Kubernetes clusters to your project that are then displayed under the **Override** column:

 **Notes:**

- If the remote clusters are not added to your project, the option **No override options** is displayed as the default value and the test runs in the Kubernetes cluster of Rational® Test Automation Server.
- If remote clusters are added to your project, the added clusters are displayed along with their availability status and ownership information.

- b. Select the location where you want to run the test asset from the following options:

- Select the **Default Cluster** when no remote clusters are available in your project.
- Select the remote cluster from the list when a remote cluster is available in your project.
- Select **No override options**, if you selected any remote cluster and want to revert to the **Default Cluster** to run the test asset.

- c. Click **Execute**.


The test run is initiated.

Results

You have configured and either started or scheduled a run of a scan of the application code that is specified in the `codesweep` file.

What to do next

You can choose to perform any of the following tasks:

- Stop the test run at any point after the test run is initiated from the **Execution** page. See [Stopping test runs on page 403](#).
- Cancel a scheduled test run from the **Execution** page. See [Canceling scheduled test runs on page 406](#).
- View all the states of the test asset by clicking the **Show in the Progress page** icon  for the test asset for which you started or scheduled the run. See [Viewing the state of test assets on page 394](#).
- View the progress of the test from the **Progress** page. See [Viewing the progress of running test assets on page 398](#).
- View the results, reports, and logs of the test from the **Results** page after the test completes the run. See [Test results on page 455](#).

Related information

[Resetting the configuration settings for a test run on page 402](#)


[Configuration of a scan of an application code on page 383](#)

[Test run configurations on page 249](#)

Management of running tests

Find information about the tasks that you can perform on a test that you configured for a run either while it runs or after it completes the run.


Viewing the state of test assets

After you initiate or schedule a test run on the **Execution** page, you can view the state of the test assets on the **Progress** page by clicking the **Show in the Progress page** icon  that is enabled for that test asset on the **Execution** page. You can view all the states of the test asset from the past hour such as `In transition`, `Running`, `Scheduled`, `Completed`, or `Stopped`.

Before you begin

You must have initiated runs of the test assets in your project from the **Execution** page.



Note: The **Show in the Progress page** icon  is enabled for the test asset only if a run of the test asset is either started or scheduled.

1. Click **Execution**, if you are not already on the **Execution** page.
2. Select the branch of the repository that contains the test assets.


The test assets that are contained in the selected branch of the repository are displayed in the following tabs on the **Execution** page:




























Tab	Description
SUITES	Lists all suites, compound tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules that are in the selected branch of the repository.
TESTS	Lists all API tests, functional tests, or performance tests that are in the selected branch of the repository.
ADVANCED	Lists all assets that are displayed when custom filters are applied for assets in the selected branch of the repository.


3. Identify the test asset or resource that you want to run by performing any of the following actions:










- Scroll through the list.


 **Tip:** You can hover over the icon in the **Type** column to know the type of the test asset.

 **Note:** You can also identify the type of the asset from the icon that represents the test type as shown in the following table:


Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the ADVANCED tab
	API test			
	Functional test			
	Performance test			
	AFT Suite			
	API Suite			
	Compound Test			
	HCL AppScan CodeSweep			
	JMeter Test			
	JUnit Test			



Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the AD-VANCED tab
	Postman test			
	Rate Schedule			
	VU Schedule			

- Search for the test asset or resource by entering any text contained in the test asset or resource name in the **Search** field.
- Click the **Filter** icon  in the **SUITES** or **TESTS** tab to filter the displayed assets based on the asset type.

For example, select **API Suite** in the **SUITES** tab to display all API Suites or select **Functional Test** in the **TESTS** tab to display all functional tests in the selected branch of the repository.

- Click the **Filter** icon  in the **ADVANCED** tab, and then create a filter query by using the **New filter** option by performing the following steps:
 - Click **New filter**.
 - Enter a name for the filter.
 - Select an operator, and add a rule, or a group of rules.
 - Add or enter the relevant parameters and either select or enter the condition and the criteria for the condition.

You can select a parameter from the following list:

- Type
 - Test Asset Name
 - Test Asset Path
 - Last Result
 - Next Run
 - Components
- Save the filter query to save and apply the filter query to filter the assets based on the query.

The test assets that match the filter criteria are displayed.

- Retrieve and apply a saved filter query, if you have saved filter queries previously by performing the following steps:



Note: The filter query applied previously is selected and the assets based on that filter query are displayed. To apply a different filter query, you must have created and saved the filter query.

- a. Click the **Filter** icon  in the **ADVANCED** tab.

The filter queries that you created and saved are displayed.

- b. Click the filter that you want to apply.

The test assets that match the filter criteria are displayed.

4. Click the **Show in the Progress page** icon  in the **Actions** column of the test asset.

The **Progress** page is displayed with the following details:

- The different states of the test asset runs that were initiated or completed in the past hour are displayed as follows:
 - In transition
 - Running
 - Completed
 - Stopped by User
 - Canceled
- The ID of the selected test asset is applied as a filter and is displayed in the filter row.

The applied filter is removed along with the displayed items for this filter when you perform any of the following actions on the **Progress** page:


- You click **Hide inactive**, and then click **Show all**.
- You visit any other page in the Navigation pane and return to the **Progress** page.

Results

You have viewed all the states of a specific test asset runs that were initiated during the past hour.

What to do next

You can choose to perform any of the following tasks from the **Progress** page:

- Stop test assets that are running. See [Stopping test runs on page 403](#).
- Cancel scheduled runs. See [Canceling scheduled test runs on page 406](#).
- View the results or the execution log of the test asset from the **Actions** column by clicking the **Open action menu** icon . See [Checking logs on page 401](#).
- View the results, reports, and logs of the completed test run from the **Results** page. See [Test results on page 455](#).

Viewing the progress of running test assets

After you initiate or schedule a test run on the **Execution** page, you can view the progress of the test assets that are running or scheduled to run, from the **Progress** page. You can also view the test assets that have completed their run, are stopped or canceled, during the past hour.

Before you begin

You must have initiated runs of the test assets in your project from the **Execution** page.

1. Open your project and click **Progress**.

Any one of the following views is displayed:

- No test assets are displayed in the **Progress** page if there are no test assets that are running, scheduled to run, or have completed their run in the past hour.
- Test assets in your project that are in any of the following states are listed:
 - Running
 - Scheduled
 - Completed
 - Stopped
 - Canceled
- No test assets are displayed but the option **Show all** is enabled. You can click **Show all** to display the test assets that ran in the past hour. Test assets that were hidden from the display when **Hide inactive** was clicked are displayed.



Note: Test assets are automatically removed from the display after *60 minutes* of being added to the **Progress** page. You can hide the test assets from the display by clicking **Hide inactive**, so that you can view only the test assets that were added to the **Progress** page in the past hour. You cannot use the **Progress** page to view the history of test assets that were run from the **Execution** page.

2. Identify the test asset by performing any of the following actions:

- Identify the test asset or resource by scrolling through the list.
- Search for the test asset by entering any text contained in the test asset name in the **Search** text box.
- Create a filter query by using the **New filter** option by performing the following steps:
 - a. Click **New filter**.
 - b. Select an operator, and add a rule, or a group of rules.
 - c. Add or enter the relevant parameters and either select or enter the condition and the criteria for the condition.

You can select a parameter from the following list:

- Type
- Name
- Started by


- Started on
 - Finished on
 - Status
 - Verdict
- d. Apply the filter query to filter the assets based on the query.

The test assets that match the filter criteria are displayed.

- e. Save the filter query by performing the following steps, if you want to reuse the filter query later:
- i. Click **Save**.
 - ii. Enter a name for the filter query.
 - iii. Click **Save**.
- Retrieve a saved filter, if you have saved filter queries earlier by performing the following steps:



Note: To open the filter query, you must have created and saved a filter query.




- a. Click the **Open filters** icon .
- b. Select the saved filter in the **Filters** dialog.
- c. Click **Apply** to apply the filter.

The test assets that match the filter criteria are displayed.

- Sort the test assets by the user who started the test by clicking the **Started by** column header, and find the test asset listed against your name.
- Sort the test assets by the type of the test assets by clicking the **Type** column header, and find the test asset type you ran or scheduled.
- Click **Hide inactive** to only display the test assets that are running or scheduled to run. This action removes all completed, canceled, stopped, or failed test assets from the display.



Note: The items in the rows are sorted in an ascending or descending order when the column header is clicked. The following icons show the status of sorting in a column:

-  items in the column are not sorted.
-  items in the column are sorted in the ascending order or with the earliest record on top.
-  items in the column are sorted in the descending order or with the latest record on top.

You can view the following details of the test assets on the **Progress** page:

Column header	Description
Type	Displays the icon for the type of the test asset.

Column header	Description
Name	<p>Displays the name of the test asset added to the project from the Git repository.</p> <p>Clicking the test name displays the Details panel. You can view the details of the test such as the description, the branch in the repository that contains the asset, the Git details, and the details of the commits to the repository. You can also view the history of the test runs for the specific test under the History tab of the Details panel.</p>
Started by	Displays the name of the user who started the run of the test asset.
Started on	Displays the date and time when the test run was started.
Finished on	Displays the date and time when the test run completed the run.
Status	<p>Displays the state of the test asset progressively.</p> <p>For example, after the run is in the <code>Initiated</code> state, it moves on to the <code>Running</code> state and ends in its final state depending on the verdict of the test run as either <code>Completed</code>, <code>Failed</code>, or <code>Inconclusive</code>.</p> <p>If the test run is stopped or a scheduled run is canceled, the status displayed is <code>Stopped by User</code> for a stopped run or <code>Canceled</code> for a canceled test.</p>

From the **Actions** column, you can perform the following operations on your test asset:

- Stop a run.
- Cancel a scheduled run.
- Cancel scheduled recurring test runs.
- View the test log.
- View the result of the test asset on the **Results** page.
- Query the test result data.


Results

You have viewed the progress of the test assets that you ran from the **Execution** page.

What to do next

You can choose to perform any of the following tasks from the **Progress** page:

- Stop test assets that are running. See [Stopping test runs on page 403](#).
- Cancel scheduled runs. See [Canceling scheduled test runs on page 406](#).

- View the results or the execution log of the test asset from the **Actions** column by clicking the **Open action menu** icon . See [Checking logs on page 401](#).
- View the results, reports, and logs of the completed test run from the **Results** page. See [Test results on page 455](#).
- Query the test results data. See [Test log queries on page 484](#).

Checking logs

To verify how the test ran or to debug test run failure, you can check the `Test Log` and `Execution log`.

About this task

The `Test Log` displays the interaction between Rational® Test Automation Server and the application or system under test. After the test run completes, the verdict of the run can be *Pass*, *Fail*, or *Inconclusive*. If the verdict of the run is *Pass*, the `Test Log` is available in the **Results** page.



The screenshot shows a test run summary for 'cmptest-1'. The test path is 'WebUIProjectCustomCode/CompoundTests/cmptest-1.testsuite', the user is 'uatester', and the duration is '00:02:28'. The status is 'Complete' with a green progress bar and a 'Pass' verdict. Below the summary, there are two panels: 'DETAILS' and 'Reports'. The 'DETAILS' panel shows 'Git Repo' as 'git@github.com:IBM/ibm-test-server-sample.git', 'Git Branch' as 'master', and 'Git Path' as 'WebUIProjectCustomCode/CompoundTests/cmptest-1.testsuite'. The 'Reports' panel contains three links: 'Statistics Report', 'Test Log', and 'Mobile and Web UI Report'.

The `Execution log` displays the console messages of the run time process that runs the test. This log is useful in determining the cause of the failure if the verdict of the run is *Fail* or *Inconclusive*. You can view the `Execution log` from the **Progress** page.

Viewing the Test Log

1. Go to the **Results** page and identify the test that you ran.
2. Click the test so that the **Reports** panel is displayed.
3. Click the **Test Log** in the **Reports** panel.

The `Test Log` is displayed in a browser window.

Viewing the Execution log

4. Go to the **Progress** page and identify the test that is in the *Running* state.
5. Click the **Open action menu** icon.
6. Click **Execution log**.

The `Execution log` is displayed in a browser window.

Related information

[Test run configurations on page 249](#)

Resetting the configuration settings for a test run

When you are configuring a test run either as a first-run or when repeating the run, and you do not want to proceed with the settings configured or saved for the test run, you can reset the configuration settings. Resetting the configuration reverts all the settings to their default values for the test run.

Before you begin

You must be a member of the project with the *Owner* or *Tester* role to run the tests.

You must ensure that you have the **Execute test asset** dialog is displayed before you proceed with resetting the configuration settings for the test run.

About this task

You can reset the configuration settings when you are configuring a test run either for its first run or when you are repeating the test run.

1. Click **Reset** in the **Execute test asset** dialog at any point when you want to reset the configuration settings for the test run.



Notes:

- If you have configured some or all of the settings for the current test run and you do not want to continue with those settings, clicking **Reset** resets the settings to their default values.
- If you are repeating a test run and do not want to use the saved settings from a previous run, clicking **Reset** reverts all the saved settings to their default values.

The settings in the **Execute test asset** dialog are reset to their default values.

2. Use the following table to find the default values for each of the settings in the **Execute test asset** dialog:

Window or Tab	Field or Setting	Default value is
Execute test asset	Scheduling the test run	Now is selected.
Execute test asset	Label for settings	Null or empty field.
Advanced	JVM Arguments	Null or empty field.
	Program Arguments	Null or empty field.
	Environment Variables	Null or empty field.

Window or Tab	Field or Setting	Default value is
ENVIRONMENT tab	API test environment	The environment configured in the test asset.
	Secrets collection	Null or empty field.
DATA SOURCES tab	Override list	Null or empty field if the asset has no dataset.
		The dataset in the test asset if the asset has one dataset.
		The first dataset on the list of datasets, which are listed in the increasing alphabetical order, if the asset has multiple datasets.
VARIABLES tab		Null or empty field.
LOCATION tab	Agents	Null or empty field if the asset has no agents configured.
		The first agent on the list of agents, which are listed in the increasing alphabetical order, if the asset has multiple agents configured.
	Docker host	Internal Docker host if no remote Docker hosts are added.
		No override options if other Docker hosts are saved in previous runs.

What to do next

You can complete configuring the settings that you want for the test run. See [Test run configurations on page 249](#).

Stopping test runs

You might want to stop a test run or multiple tests that are running when you realized that you did not configure all the settings or you want to change a few settings for the test runs.

Before you begin


You must have initiated a run for one or multiple test assets from the **Execution** page.

About this task



You can stop either a running test or multiple running tests from the **Progress** page. You can stop the test if it is in either the `In transition` or `Running` state. You cannot stop the test run if the test has already completed its run.

1. Go to the **Progress** page and identify the tests in any of the following ways that you want to stop:

You can select the tests that satisfy the following conditions:

- The test must be either in the *In transition* OR *Running* state.
- The **Stop execution** icon  in the **Actions** column must be enabled for the test.
- The checkbox must be enabled for the test.


2. Perform the actions in the following table:

If...	Action...
<p>You want to stop any test that is running</p>	<p>Perform any of the following actions:</p> <ul style="list-style-type: none"> ◦ Click the Stop execution icon  in the Actions column of the selected test. ◦ Click the checkbox in the row of the selected test, and then click Stop selected. <p>The Stop execution dialog is displayed.</p>
<p>You want to stop multiple tests that are running</p>	<p>Perform any of the following actions:</p> <ul style="list-style-type: none"> ◦ Click the checkbox in the row of the selected tests, and then click Stop selected. ◦ Click the Select all checkbox in the header row, and then click Stop selected. <p> Note: All running tests are selected when you click the Select all checkbox.</p> <p>The Stop execution dialog is displayed.</p>

3. Perform the following steps in the **Stop execution** dialog that is displayed:

- a. Set the timeout period for stopping the test run. Enter a numeric value and select the unit from the options available as *Seconds*, *Minutes*, or *Hours*.

The time out period is the time during which the test run is allowed to stop on its own and after the timeout period, the test is forced to stop abruptly.

 **Important:** If the timeout period is not set, a value of *30 seconds* is considered as the default timeout period.

- b. Perform any of the following actions:

- Keep the **Capture results** option selected, if you want the results to be captured for the test.

The captured results for the test are available to view from the **Results** page.

- Clear this option if you do not want to capture the results.



Note: The **Capture results** option is selected as the default option.



Important: The results are always captured for API suites even if you clear the **Capture results** option.

c. Perform any of the following actions:

- Keep the **Execute 'finally-block', if present** option selected, if you want the *finally-block* code in the test script, if present, to be run before the test run is stopped.
- Clear this option if you do not want to run the *finally-block* code in the test script.



Note: The option **Execute 'finally-block', if present** is selected as the default option.



Restriction: The *tear-down* steps equivalent to the **'finally-block'** code, if present in API suites are not controllable at test runtime. The **Execute 'finally-block', if present** option has no impact on API suites.

d. Click **Stop execution**.

A notification is displayed that the running tests are stopped.




Note: If the test run completes before you can configure the options for stopping the test run, a notification is displayed in the **Stop execution** dialog that the test asset has completed its run.

Results

You have successfully stopped a running test or stopped multiple running tests. The stopped test runs are displayed on the **Progress** page with the status as *Stopped by User*. Stopped tests are not considered in the count of tests executed that are displayed on the **Overview** page.

What to do next

You can re-initiate the test run from the **Execution** page or from the **Results** page by completing the configurations that you want for the test.

You can view the results or the execution log of the stopped test from the **Actions** column on the **Progress** page by clicking the **Open action menu** icon .

Related information

[Test run configurations on page 249](#)

[Viewing the progress of running test assets on page 398](#)

[Canceling scheduled test runs on page 406](#)

Canceling scheduled test runs

You might want to cancel a scheduled test run, scheduled recurring test runs, or multiple scheduled test runs when you realized that you did not configure all the settings for the runs, want to change a few settings, or do not want those tests to run.

Before you begin

You must have scheduled runs for one or multiple test assets from the **Execution** page.

About this task

You can cancel a scheduled test run, all instances of a scheduled recurring test runs, or multiple scheduled tests from the **Progress** page.

1. Perform any of the following actions:


- Click **Scheduled** under the **Next Run** column on the **Execution** page.

The test is displayed on the **Progress** page. If you selected a test that was scheduled as a recurring test, then all the tests that ran and the next scheduled test run are displayed.

Go to [Step 2 on page 406](#).

- Go to the **Progress** page and identify the tests in any of the following ways that you want to cancel:



You can select the tests that satisfy the following conditions:

- The test must be in the `Scheduled` state.
- The **Cancel** icon  in the **Actions** column must be enabled for the test.
- The checkbox must be enabled for the test.

Go to [Step 2 on page 406](#).

2. Perform the actions in the following table:

If...	Action...
You want to cancel any test run that is scheduled.	Perform any of the following actions:

If...	Action...
You want to cancel all scheduled instances of the recurring test runs.	<ul style="list-style-type: none"> ◦ Click the Cancel icon  in the Actions column of the test that displays <i>Scheduled</i> as the status. ◦ Click the checkbox in the row of the test that is scheduled, and then click Stop selected. <p>The Cancel scheduled execution dialog is displayed.</p> <p>Go to Step 3 on page 407.</p>
You want to stop multiple test runs that are scheduled	<p>Perform any of the following actions:</p> <ul style="list-style-type: none"> ◦ Click the checkbox in the row of the selected tests, and then click Stop selected. ◦ Click the Select all checkbox in the header row, and then click Stop selected. <p> Note: All scheduled tests are selected when you click the Select all checkbox.</p> <p>The Cancel scheduled execution dialog is displayed.</p> <p>Go to Step 3 on page 407.</p>

3. Click **Yes** in the **Cancel scheduled execution** dialog.

Result

.

A notification is displayed that the scheduled test runs are canceled.

Results

You have successfully canceled a scheduled test, all instances of the scheduled recurring tests, or multiple scheduled tests. The canceled test runs are displayed on the **Progress** page with the status as *Canceled*. The canceled test runs are not displayed on the **Results** page and are not considered in the count of tests executed that are displayed on the **Overview** page.

What to do next

You can re-initiate the test run from the **Execution** page by completing the configurations that you want for the test.

Related information

[Test run configurations on page 249](#)

[Viewing the progress of running test assets on page 398](#)

[Stopping test runs on page 403](#)

Management of virtualized services



Disclaimer: This release contains access to the virtual services that virtualize the Istio based services feature in IBM® Rational® Test Automation Server as a Tech Preview. The Tech Preview is intended for you to view the capabilities for virtualized services offered by Rational® Test Automation Server, and to provide your feedback to the product team. You are permitted to use the information only for evaluation purposes and not for use in a production environment. IBM provides the information without obligation of support and "as is" without warranty of any kind.

You can find information about the tasks that you can perform on and manage the virtualized services that run on Rational® Test Automation Server. You can start or stop the virtual services that are connected to Rational® Test Automation Server. You can view the routing rules and usage statistics of virtualized services, agents, or intercepts that are connected to Rational® Test Automation Server.

You must create stubs or virtual services in Rational® Integration Tester for the following types of services. You must commit the virtual service resources to a remote repository and then add the repository to your project on Rational® Test Automation Server before you can run the virtual services on Rational® Test Automation Server:

- Virtual services that utilize the WebSphere® MQ transport.
- Virtual services that utilize the HTTP transport.
- Virtual services that virtualize the Istio based services. You can run virtual services for the following types of requests received or sent by the Istio service mesh:
 - Requests received by services in the Istio service mesh.
 - Requests sent from namespaces in the Istio service mesh to external services that are not in the Istio service mesh.



Important: You can run virtual services only in the **Default Cluster** location of Rational® Test Automation Server. You cannot run virtual services on a remote Docker host.

The following are the considerations that decide how a container of the Rational® Test Automation Server cluster is selected to run virtual services:

- Before a container is selected to run a virtual service, it is checked for a previous failure to start a virtual service or for a low heap size, and other conditions. If the container was not found suitable, then it is not selected for running the virtual service.
- Multiple virtual services can be run in the same container if the following conditions are satisfied:
 - Subsequent virtual services are started by the same project member.
 - All virtual services are created and contained in the same Rational® Integration Tester project.
 - The requests to run the virtual services refer to the same version, meaning that the git commit IDs on which they are based are the same. If you use the UI, this means that the **Version** field in the **Execute virtual service** dialog are the same.
 - The settings that you configure in the **Advanced** settings for the virtual services are the same.
 - The datasets selected for an override are the same, if datasets are used.

Working with virtual services

You can perform the following tasks on virtual services on Rational® Test Automation Server:

- Read about the considerations before you configure a run of the virtual services. See [Prerequisites for running virtual services on page 410](#).
- Read about the considerations before you configure a run of the Istio based services that are virtualized. See [Prerequisites for running virtualized Istio based services in IBM Rational Test Automation Server on page 412](#).
- Set up the HTTP proxy, if you have configured stubs in the test assets to use an HTTP proxy to route requests. See [Setting up the HTTP proxy on page 413](#).
- View all intercepts that are registered with Rational® Test Automation Server including those that you have not configured. See [Viewing intercepts that are registered with a team space on IBM Rational Test Automation Server on page 415](#).
- View virtual service resources in the test assets that are in the repositories added to a project on Rational® Test Automation Server. See [Viewing virtual service resources on page 416](#).
- Configure runs of virtual services. See [Configuring a run of a virtual service on page 419](#).
- Configure runs of HTTP virtual services to run without using proxies. See [Running HTTP virtual services without using proxies on page 433](#).
- Run Istio stubs that were created in Rational® Integration Tester to virtualize services in the Kubernetes cluster on which Rational® Test Automation Server is installed.
- View running instances of virtual services. See [Viewing running instances of virtual services on page 438](#).
- View configurations of a running virtual service instance. See [Viewing configurations of running instances of virtual services on page 443](#).
- Modify the configurations of a running virtual service instance. See [Modifying configurations of running instances of virtual services on page 447](#).
- View details of the usage statistics of the virtual services. See [Viewing usage statistics of virtual services on page 451](#).
- View routing rules of the intercepts. See [Viewing routing rules of the virtual services on page 449](#).
- Stop a running virtual service instance. See [Stopping virtual services on page 454](#).

Prerequisites for running virtual services


You can find information about the prerequisite tasks that you must complete before you configure a run of the virtual services on IBM® Rational® Test Automation Server.

Prerequisites for running HTTP virtual services

You can find information about the tasks that you must complete before you configure a run of the HTTP virtual services on IBM® Rational® Test Automation Server.

Conditions for running of HTTP virtual services

Before you run the HTTP virtual services from Rational® Test Automation Server, you must check for the following conditions and perform the actions indicated:

If...	Then...
<p>You want to route the HTTP traffic via the HTTP proxy either to the virtual service or live systems, based on routing rules.</p>	<p>You must set up the HTTP proxy. See Setting up the HTTP proxy on page 413.</p>
<p>You want to route the HTTP traffic to the virtual service via the HTTP proxy and the proxy is not configured to trust all certificates.</p>	<p>The trust store must be configured to include the certificate authority for the server certificate. See Certificate authority: Importing and extending lists on page 115.</p>
<p>A client application is sending requests directly to the virtual service.</p>	<p>The client application must trust the certificate authority for the server certificate. See Certificate authority: Importing and extending lists on page 115.</p>
<p>You are working with Rational® Test Automation Server V10.1.3, which is installed on Ubuntu.</p>	<p>HTTP virtual services are exposed via host names are of the following form:</p> <pre data-bbox="836 1409 1414 1444">in-<code><unique_id></code>.<INGRESS_DOMAIN></pre> <p> Note: The DNS used by a computer that runs a client application or the HTTP proxy that routes traffic to the virtual service needs to resolve such host names to the IP address of the ingress domain.</p> <p>You must perform the following actions that depend on the method you have used to set up the ingress domain:</p>

If...	Then...
	<ul style="list-style-type: none"> • If you used <code>nip.io</code> to form the ingress domain, then that virtual services host names are automatically resolved to the IP address of the ingress domain. For example, if you used <code>nip.io</code> to form the ingress domain as follows: <pre>10.1.2.3.nip.io</pre>The virtual services host names are resolved by <code>nip.io</code> to the IP address as <code>10.1.2.3</code>. • If your ingress domain does not use <code>nip.io</code> then you must ensure that the DNS in use by the client application or the HTTP proxy can resolve host names of the virtual services to the IP address of the ingress domain. For example, if the ingress domain is as follows: <pre>myhost.mycom.com</pre>The DNS must resolve <code>*.myhost.mycom.com</code> to the IP address of <code>myhost.mycom.com</code>.

HTTPS virtual service endpoints

When you want to run an HTTP virtual service where the HTTP transport settings have **Use SSL** enabled, the virtual service endpoint that is exposed for the virtual service instance expects an HTTPS request on port 443. The exposed endpoint is managed by the same ingress controller or the HAProxy that is used for the server endpoints. The SSL/TLS termination occurs at the endpoint before the request is routed to the virtual service instance and results in the following implications:

- The **Virtualization (Server) SSL** options that are specified in the HTTP transport have no effect.
- The certificate served by the exposed virtual service endpoint is the same as that of server that is signed by the server certificate authority.

Collection of usage metrics

If you want Rational® Test Automation Server to display the usage statistics of the virtual service instances, you must install the Prometheus server at the time of server installation.

Related information

[Setting up the HTTP proxy on page 413](#)

[Running HTTP virtual services without using proxies on page 433](#)

Prerequisites for running virtualized Istio based services in IBM® Rational® Test Automation Server



Disclaimer: This release contains access to the virtual services that virtualize the Istio based services feature in IBM® Rational® Test Automation Server as a Tech Preview. The Tech Preview is intended for you to view the capabilities for virtualized services offered by Rational® Test Automation Server, and to provide your feedback to the product team. You are permitted to use the information only for evaluation purposes and not for use in a production environment. IBM provides the information without obligation of support and "as is" without warranty of any kind.

You can find information about the tasks that you must complete before you run virtualized Istio based services in Rational® Test Automation Server.

Configuring virtualized Istio based services in Rational® Test Automation Server

When you intend to virtualize services in the Istio mesh in Rational® Test Automation Server, you can create stubs in Rational® Integration Tester.

When you create the Istio stubs in Rational® Integration Tester, you must ensure that you are aware of the fully qualified domain name (FQDN) or the short name of the service that you want to virtualize in the Kubernetes cluster that hosts Rational® Test Automation Server.

The FQDN of the service is as follows:

```
<service-name>.<namespace>.<svc>.<cluster-name>
```

The short name of the service is as follows:

```
<service-name>
```

For example, if you want to virtualize the HTTP traffic in the service named as *reviews* that runs in a namespace named as *bookinfo*, and in the resource named as *svc* in the Kubernetes cluster named as *mycluster*, then the FQDN of the service is as follows:

```
reviews.bookinfo.svc.mycluster
```

The short name of the service is as follows:

```
reviews
```

You must enter the host of the service that you want to virtualize, in the **Host** field of the physical transport in Rational® Integration Tester. You can provide either the FQDN or the short name of the service.



Note: The pass-through action in the stubs is not supported.

Considerations for running of stubs that virtualize Istio services

Before you run the stubs that virtualize services in namespaces contained in the Rational® Test Automation Server Kubernetes cluster, you must install Rational® Test Automation Server with the *demo* configuration.

When you install Rational® Test Automation Server by using the *demo* configuration, then Rational® Test Automation Server is preconfigured to use the **bookinfo** namespace as the virtualization namespace.



Note: You can change or add virtualization namespaces by adding the helm installation argument `execution.istio.namespaces` along with a list of namespaces in a helm command.

After you author the virtualized Istio based services, you must commit the virtual service resources to a repository and add the repository to the project on Rational® Test Automation Server. You can configure a run of the virtualized Istio service. See [Configuring a run of a virtual service on page 419](#).

Collection of usage metrics

If you want Rational® Test Automation Server to display the usage statistics of the virtual service instances, you must install the Prometheus server at the time of server installation.

Related reference

[Installation of the server software on page 29](#)

Setting up the HTTP proxy

When you want to run HTTP virtual services on IBM® Rational® Test Automation Server, you can set up an HTTP proxy to route requests to the virtual service. You must configure the HTTP proxy with the Rational® Test Automation Server URL and offline user token generated from Rational® Test Automation Server so that the HTTP proxy can register with Rational® Test Automation Server.

Before you begin

You must have completed the following tasks:

- Installed the proxy component from Rational® Test Control Panel of the same version as that of Rational® Test Automation Server.
- Read the following procedures:
 - Generating offline user tokens. See [Managing access to IBM Rational Test Automation Server on page 600](#).
 - Viewing the *Team Space ID* as the URL alias of the team space. See [Viewing the configuration of a team space on page 609](#).

About this task

You must edit the `registration.xml` file that is located in the `<RTCP_install_directory>\httptcp` directory for the following attributes:


- **server base-url**
- **security-token**
- **team-space-id**

You must enter the Rational® Test Automation Server URL, offline user token, and the ID of the team space to which the proxies must register as the values for the attributes. After you restart the proxy, the HTTP proxy registers with the team space on Rational® Test Automation Server. You can view the registered intercepts from the **Infrastructure > Agents and Intercepts** page in your team space.


1. Perform the following steps in Rational® Test Automation Server:

a. Log in to Rational® Test Automation Server.

The **Team Space Dashboard** page is displayed.

b. Generate the offline user token by clicking the **User** icon , and then click **Create Token**.

c. Copy the offline user token to a text file.

d. Click the **Settings** icon  of the team space in the **My Team Spaces** page.

The **Team Space Configuration** page is displayed.

e. Copy the text that is displayed under **URL alias** as the *Team Space ID*.

2. Perform the following steps to edit the `registration.xml` file that is located in the `<RTCP_install_directory>\httptcp` directory on the computer on which you have installed Rational® Test Control Panel:

a. Open the `registration.xml` file by using a text editor.

b. Enter the values for the following attributes:



Note: If an entry exists, you must either comment out that entry and add a new one or replace the contents with the values indicated.

Attribute	Value
server base-url	URL of Rational® Test Automation Server
security-token	Offline user token generated from Rational® Test Automation Server
team-space-id	ID of the team space to which the proxies must register

- c. Save and close the file.
 - d. Restart the HTTP proxy.
3. Perform the following steps in Rational® Test Automation Server to view the HTTP proxies that are registered in the team space:
 - a. Log in to Rational® Test Automation Server.
 - b. Click **Infrastructure > Agents and Intercepts** to view the registered intercepts.

The HTTP proxies that you configured with your offline user token and *Team Space ID* are displayed.

Results

You have successfully configured the HTTP proxies and viewed that they are registered in the team space on Rational® Test Automation Server.

What to do next

You can perform the following tasks:

- Read for the prerequisite tasks that you must perform before you can run the HTTP stubs. See [Prerequisites for running HTTP virtual services on page 410](#).
- Run the virtual service resource. See [Configuring a run of a virtual service on page 419](#).
- View the routing rules for the intercept. See [Viewing routing rules of the virtual services on page 449](#).
- View the routing rules for the intercept in a running instance of the virtual service. See [Viewing configurations of running instances of virtual services on page 443](#).

Viewing intercepts that are registered with a team space on IBM® Rational® Test Automation Server

After you install the HTTP proxies and configure them with your offline user token, the proxies register with Rational® Test Automation Server in a team space as intercepts and you can view the registered intercepts on the **Agents and Intercepts** page for the team space.

Before you begin

You must have completed the following tasks:

- Completed the prerequisite tasks before you configure a run of the HTTP virtual services. See [Prerequisites for running HTTP virtual services on page 410](#).
- Read the prerequisite information about virtualizing the Istio based services. See [Prerequisites for running virtualized Istio based services in IBM Rational Test Automation Server on page 412](#).
- Installed the HTTP proxies of the same version as the version of Rational® Test Automation Server.
- Configured the HTTP proxy with your offline user token so that the proxy can register as an intercept with Rational® Test Automation Server. See [Setting up the HTTP proxy on page 413](#).

- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Click **Infrastructure > Agents and Intercepts** in the navigation pane.

The **Agents and Intercepts** page is displayed.

You can view the agents, intercepts, or Docker hosts that are registered in the team space on Rational® Test Automation Server.

3. You can view the intercepts that you own in any of the following ways:

- Search for the intercept by entering the name of the intercept in the **Search** field.



Note: You can enter either the full name or any text that is in the name. The search is enabled for case-sensitive text that you can enter.

- Sort the **Type** column to show the items sorted with the intercepts in the rows at the top of the table.
- Sort the **Agents** column and identify the intercepts by their names or by their owner.

You can view the following details about the intercepts that are registered with Rational® Test Automation Server:

- *Any project* is displayed in the **Projects** column indicating that the intercept can be used by virtual services in any project.
- The status of the intercept is displayed in the **Status** column.

Results

You have viewed the registered intercepts from the **Agents and Intercepts** page.

What to do next

You can start the virtual services that use the registered intercepts. See [Configuring a run of a virtual service on page 419](#).

Viewing virtual service resources

When project repositories contain virtual service resources and you want to view the resources so that you can start them, you can view the virtual services from the **Resources** page in your team space on Rational® Test Automation Server.


Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created virtual services that use any of the supported transports for tests in Rational® Integration Tester, committed the test resources to the remote repository, and added the repository to your project.

About this task

After you have committed the test assets and resources to the remote repositories and added the repositories to projects in your team space on Rational® Test Automation Server, you can go to the **Resources** page and perform the following tasks:

- View the virtual service resources that are contained in any branch of the repositories that are added to your project.
- View the components in the System Model that are associated with the virtual service resources.
- Create or use saved filters for viewing virtual service resources.
- Run or start an instance of the virtual service resource.
- View the number of instances of the virtual service resources that are running.
- View all instances of a running virtual service on the **Instances** page by clicking the **Show in instances page** icon .

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the virtual service resources in the test assets by clicking **Projects > My Projects > *project_name***.




The **Overview** page is displayed.


3. Click **Virtualization > Resources** in the navigation pane.

The **Resources** page is displayed.

4. Complete the steps for the task that you want to perform as listed in the following table:

Task	Action
Viewing the virtual service resources that are contained in a branch of any of the repositories that are added to your project.	Select the branch of the repository that contains the virtual services that you want to run from the list in the Branch field. All virtual services in the selected branch are displayed on the Resources page.

Task	Action
Viewing the components that are associated with the virtual service resource.	<p>The components that are associated with the virtual service resource are displayed in the Components column.</p> <p>Click the component in the Components column to view the virtual resource on the System Model page.</p>
Creating or using saved filters for viewing specific instances of virtual service resources.	<p>To create a query, perform the following actions:</p> <ol style="list-style-type: none"> Click New filter. Create a rule with an appropriate operator. Select criteria such as <i>Name</i>, <i>Path</i>, <i>Instance number</i>, <i>Instance state</i>, or <i>Components</i>. Select the condition, and then enter the value for the criteria. Apply the filter query. <p>The virtual services that match the filter criteria are displayed.</p> <ol style="list-style-type: none"> Save the filter query for retrieving it from the saved filters list. <p>To use a saved filter query, perform the following actions:</p> <p> Note: To open the filter query, you must have created and saved a filter query.</p> <ol style="list-style-type: none"> Click the Open filters icon . Select the saved filter. Apply the filter. <p>The virtual services that match the filter criteria are displayed.</p>
Running an instance of the virtual service resource.	<p>After you identify the virtual service that you want to run, perform the following actions:</p> <ol style="list-style-type: none"> Click the Execute icon  in the row of the identified virtual service. <p>The Execute virtual service dialog is displayed.</p> <ol style="list-style-type: none"> Run the virtual service with the settings that are configured in the resource by clicking Execute or modify any of the settings, and then click Execute.

Task	Action
	<p>An instance of the virtual service starts to run. The virtual service is displayed with the state as <code>Running</code> along with the number of instances that are running in the Active instances column.</p>
<p>Viewing the number of instances of the virtual service resources that are running.</p>	<p>If any of the virtual services in the project repositories are started by any project member with the <i>Tester</i> or <i>Owner</i> role, you can view the number of instances that are running or active from the Resources page.</p> <p>The virtual services are displayed with the state as <code>Running</code> along with the number of instances that are running, in the Active instances column.</p>
<p>Viewing all instances of a running virtual service.</p>	<p>Click the Show in instances page icon  in the row of the virtual service on the Resources page.</p> <p>The Instances page is displayed with all the running instances of a particular virtual service.</p> <p>You can also view the number of requests that are received by the virtual service. The number of requests is displayed as the number of hits when you hover the cursor over the text in the Activity column.</p>

Results

You have viewed virtual services from the **Resources** page on Rational® Test Automation Server.

Related information

[Configuring a run of a virtual service on page 419](#)

Configuring a run of a virtual service



Disclaimer: This release contains access to the virtual services that virtualize the Istio based services feature in IBM® Rational® Test Automation Server as a Tech Preview. The Tech Preview is intended for you to view the capabilities for virtualized services offered by Rational® Test Automation Server, and to provide your feedback to the product team. You are permitted to use the information only for evaluation purposes and not



for use in a production environment. IBM provides the information without obligation of support and "as is" without warranty of any kind.

When you have virtual service resources that are in the repositories added to your project, you must configure a run of the virtual service to start an instance of the virtual service on Rational® Test Automation Server or other remote Kubernetes clusters.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created virtual services that use any of the supported transports for tests in Rational® Integration Tester, committed the test resources to the remote repository, and added the repository to your project.
- Completed the prerequisite tasks before you configure a run of the HTTP virtual services. See [Prerequisites for running HTTP virtual services on page 410](#).
- Read the prerequisite information about virtualizing the Istio based services. See [Prerequisites for running virtualized Istio based services in IBM Rational Test Automation Server on page 412](#).
- Read and completed the tasks mentioned in [Considerations for using a remote Kubernetes cluster on page 235](#), if you want to use a remote Kubernetes cluster as a location to run the test.

About this task

You can start the following types of stubs contained in API Suites from the **Resources** page on Rational® Test Automation Server:

- Virtual services that utilize the WebSphere® MQ transport.
- Virtual services that utilize the HTTP transport.
- Virtual services that virtualize the Istio based services. You can run virtual services for the following types of requests received or sent by the Istio service mesh:
 - Requests received by services in the Istio service mesh.
 - Requests sent from namespaces in the Istio service mesh to external services that are not in the Istio service mesh.

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the virtual service resources in the test assets by clicking **Projects > My Projects > *project_name***.

The **Overview** page is displayed.

3. Click **Virtualization > Resources** in the navigation pane.

The **Resources** page is displayed.

4. Select the branch of the repository that contains the virtual services that you want to run from the list in the **Branch** field.

All virtual services in the selected branch are displayed on the **Resources** page.

5. Identify the virtual service by completing any of the following steps:
 - Search for the virtual service by entering the name or the path of the virtual service in the repository in the **Search** field box.
 - Create a filter query by using the **New filter** option and complete the following steps:
 - a. Create a rule with an appropriate operator.
 - b. Select criteria such as *Name*, *Path*, *Instance number*, or *Instance state*. Select the condition and enter the value for the criteria. Apply the filter query.

The virtual services that match the filter criteria are displayed.



- c. Save the filter query for retrieving it from the saved filters list.
- Retrieve a saved filter by using the **Open filters** icon  by completing the following steps:



Note: To open the filter query, you must have created and saved a filter query.

- a. Select the saved filter.
- b. Apply the filter.

The virtual services that match the filter criteria in the filter that is applied are displayed.

6. Click the **Open action menu** icon  in the row of the identified virtual service, and then click **Execute** icon .

The **Execute virtual service** dialog is displayed.

7. Select the version of the virtual service that is in the repository that you want to start by performing any of the following actions:

- Expand the list in the **Version** field, find the version of the test resources, and then select the version.

Use the following details about the version of the test resources that are displayed to identify the version that you want:


- Commit message.
- Tags labeled for the version committed.
- The user who committed the version to the repository.
- Relative time of the commit. For example, *2 hours ago* or *3 days ago*.

The list displays the versions of the test resources committed by all users to the branch in the repository. The versions are arranged with the latest version committed followed by the versions committed previously.

- Expand the list in the **Version** field, and then search for the version that you want to select by entering a partial or the complete commit message of that version.

The version that matches the search criteria is displayed and it is selected for the test run.


8. Select the environment that was used to bind the physical and logical resource in the API project, in the **ENVIRONMENT** tab.

 **Important:** The configuration that you set for the run in the **Execute virtual service** dialog is preserved when you run the same virtual service again. The configurations that you set are not available to other members when they want to run the virtual service. For example, if you selected an environment, the same environment is selected when you run the virtual service again.

9. Enter a label, if required.

A label that you enter for the test run that helps you to identify the virtual service instance on the **Instances** page. The label that you entered is displayed for the virtual service under the **Labels** column on the **Instances** page. After you have created a label, any member of the project can use that label.

10. Follow the instructions if you want to modify the configurations for the behavior of the virtual service:
 - a. Click the **BEHAVIOR** tab, if it is not already open.
 - b. Configure or change the settings for the following options:

 **Note:** The settings displayed are the settings that were configured for the virtual service when it was authored in Rational® Integration Tester.

Option	Description				
Performance	<p>The following settings are available for handling of requests by the virtual service:</p> <table border="1" data-bbox="574 1539 1417 1869"> <thead> <tr> <th data-bbox="574 1539 889 1602">Option</th> <th data-bbox="889 1539 1417 1602">Description</th> </tr> </thead> <tbody> <tr> <td data-bbox="574 1602 889 1869">Optimize performance</td> <td data-bbox="889 1602 1417 1869">If enabled, attempts are made to reduce the amount of processing between the time the virtual service receives a request and the time it sends a response. Specific optimizations depend on the message contents, as in the following examples:</td> </tr> </tbody> </table>	Option	Description	Optimize performance	If enabled, attempts are made to reduce the amount of processing between the time the virtual service receives a request and the time it sends a response. Specific optimizations depend on the message contents, as in the following examples:
Option	Description				
Optimize performance	If enabled, attempts are made to reduce the amount of processing between the time the virtual service receives a request and the time it sends a response. Specific optimizations depend on the message contents, as in the following examples:				

Option	Description									
	<table border="1"> <thead> <tr> <th data-bbox="573 260 889 308">Option</th> <th data-bbox="889 260 1425 308">Description</th> </tr> </thead> <tbody> <tr> <td data-bbox="573 308 889 930"></td> <td data-bbox="889 308 1425 930"> <ul style="list-style-type: none"> ▪ When the virtual service receives requests, all validations are disabled, and for all XML payloads, the store and filter actions are converted to use XPath expressions. ▪ When the virtualization sends responses, any store actions set on a message are disabled, and any XML content is collapsed when the virtual service is compiled instead of being collapsed every time that a response is sent. <p>The optimization is disabled as the default action for this setting and you can enable the performance optimization if you fully understand the implications of optimizing the performance.</p> </td> </tr> <tr> <td data-bbox="573 930 889 1108">Threads</td> <td data-bbox="889 930 1425 1108"> <p>The maximum number of threads used in processing requests received by the virtual service. The default number of threads is <i>10</i>.</p> </td> </tr> </tbody> </table>	Option	Description		<ul style="list-style-type: none"> ▪ When the virtual service receives requests, all validations are disabled, and for all XML payloads, the store and filter actions are converted to use XPath expressions. ▪ When the virtualization sends responses, any store actions set on a message are disabled, and any XML content is collapsed when the virtual service is compiled instead of being collapsed every time that a response is sent. <p>The optimization is disabled as the default action for this setting and you can enable the performance optimization if you fully understand the implications of optimizing the performance.</p>	Threads	<p>The maximum number of threads used in processing requests received by the virtual service. The default number of threads is <i>10</i>.</p>			
	Option	Description								
	<ul style="list-style-type: none"> ▪ When the virtual service receives requests, all validations are disabled, and for all XML payloads, the store and filter actions are converted to use XPath expressions. ▪ When the virtualization sends responses, any store actions set on a message are disabled, and any XML content is collapsed when the virtual service is compiled instead of being collapsed every time that a response is sent. <p>The optimization is disabled as the default action for this setting and you can enable the performance optimization if you fully understand the implications of optimizing the performance.</p>									
Threads	<p>The maximum number of threads used in processing requests received by the virtual service. The default number of threads is <i>10</i>.</p>									
Operation	Specifies an operation referenced by the virtual service.									
Response time	<p>Specifies the response time behavior for responses sent by the virtual service for the selected operation.</p> <p>You can modify the value by selecting a value from the following options:</p> <table border="1" data-bbox="573 1371 1425 1822"> <thead> <tr> <th data-bbox="573 1371 727 1428">Option</th> <th data-bbox="727 1371 1425 1428">Description</th> </tr> </thead> <tbody> <tr> <td data-bbox="573 1428 727 1522">No delay</td> <td data-bbox="727 1428 1425 1522">Select this option for a response with no delay.</td> </tr> <tr> <td data-bbox="573 1522 727 1654">Minimum delay</td> <td data-bbox="727 1522 1425 1654">Select this option for a delay in the response by entering the required delay in milliseconds (ms).</td> </tr> <tr> <td data-bbox="573 1654 727 1822">Uniform distribution</td> <td data-bbox="727 1654 1425 1822">Select this option for a uniformly distributed response time by specifying the minimum and maximum delay in milliseconds (ms).</td> </tr> </tbody> </table>		Option	Description	No delay	Select this option for a response with no delay.	Minimum delay	Select this option for a delay in the response by entering the required delay in milliseconds (ms).	Uniform distribution	Select this option for a uniformly distributed response time by specifying the minimum and maximum delay in milliseconds (ms).
Option	Description									
No delay	Select this option for a response with no delay.									
Minimum delay	Select this option for a delay in the response by entering the required delay in milliseconds (ms).									
Uniform distribution	Select this option for a uniformly distributed response time by specifying the minimum and maximum delay in milliseconds (ms).									

Option	Description									
	Option	Description								
	Gaussian distribution	Select this option for a response time with Gaussian distribution by specifying the minimum and maximum delay in milliseconds (ms).								
Passthrough	<p>Specifies the pass through behavior for the selected operation when requests are not handled within the virtual service.</p> <p>You can modify the value by selecting a value from the following options:</p> <table border="1" data-bbox="574 695 1422 1029"> <thead> <tr> <th data-bbox="574 695 721 835">Option</th> <th data-bbox="721 695 1422 835">Description</th> </tr> </thead> <tbody> <tr> <td data-bbox="574 695 721 835">Discard</td> <td data-bbox="721 695 1422 835">This option stops the system under test from receiving the intercepted message. This option can disrupt the calling system. For example, the system might time out while it waits for a reply.</td> </tr> <tr> <td data-bbox="574 835 721 932">Pass Through</td> <td data-bbox="721 835 1422 932">This option passes the intercepted message to the system under test, with an optional delay.</td> </tr> <tr> <td data-bbox="574 932 721 1029">Simulate Error</td> <td data-bbox="721 932 1422 1029">This option returns an error to the calling system. The message is not passed to the system under test.</td> </tr> </tbody> </table>		Option	Description	Discard	This option stops the system under test from receiving the intercepted message. This option can disrupt the calling system. For example, the system might time out while it waits for a reply.	Pass Through	This option passes the intercepted message to the system under test, with an optional delay.	Simulate Error	This option returns an error to the calling system. The message is not passed to the system under test.
Option	Description									
Discard	This option stops the system under test from receiving the intercepted message. This option can disrupt the calling system. For example, the system might time out while it waits for a reply.									
Pass Through	This option passes the intercepted message to the system under test, with an optional delay.									
Simulate Error	This option returns an error to the calling system. The message is not passed to the system under test.									



11. Follow the instructions, if the virtual service references datasets.

- You can use the dataset referenced in the asset.
- You can choose to override the dataset with another data source. If alternative data sources are available, select from the set of overrides available.

12. Follow the instructions if the virtual service requires a variable that must be passed at run time.

a. Click the **VARIABLES** tab, if it is not already open.

b. Choose one of the following methods to add the variables:

- To add new variables manually, click the **Add Variable** icon , enter the name, and value of the variable.
- To add new variables from your local computer or from the Git repository that is associated with your server project, click the **Upload** icon  and select the **Upload from local system** or **Browse from server** to select the variable file.



Note: You must have created a file with the variables before you can select the file.

13. Click **Advanced** to make the following advanced configurations:

- a. Enter any JVM arguments that must be passed at run time in the **JVM Arguments** field.



Note: Each JVM argument should be separated with white space.

For example, you can set a maximum Java heap size.

- b. Enter the environment variables that must be passed at run time in the **Environment Variables** field, if applicable.

For example, enter the environment variables when the third-party libraries that are used in the run refer to the environment variables for configuration.

- c. Select the stub logging level for the virtual service from the following options in the **Logging** list:

Option	Description
None	Specifies that the virtual service does not write log messages.
Normal	Specifies that the virtual service writes informational messages.
Debug	Specifies that the virtual service writes informational and debugging messages.

- d. Enter other configuration options as parameters and their values in the **Additional Configuration Parameters** fields, if applicable.

You can refer to the additional parameters that you can use for virtual services from the topic in the related links.

For example, if you want to start the virtual service to run in a new container, you can specify the following parameter and its value:

Parameter name	Value
<code>stub.dedicated.container</code>	<code>true</code>



Note: Click **Add** to add additional parameters.



Note: You must separate the arguments or variables with a white space when you enter them in the same line or start each argument or variable on a new line.

The default value for the fields for the advanced settings is null or an empty field.



Notes:



- If you have configured some or all of the settings for the current run, and you do not want to continue with those settings, you can reset the settings by clicking **Reset**.
- If you want to repeat a run and do not want to use the saved settings from a previous run, you can reset all the saved settings to their default values by clicking **Reset**.

14. Follow the instructions if you want to change the location of a Kubernetes cluster for running the test:

- a. Click the **LOCATION** tab, if it is not already open.

The **Default Cluster** is the default location where the virtual service resource runs, and it is listed under the **Host** column. The information about the availability of the default location is displayed.



Important: You must have added the registered remote Kubernetes clusters to your project that are then displayed under the **Override** column:



Notes:

- If the remote clusters are not added to your project, the option **No override options** is displayed as the default value and the virtual service resource runs in the Kubernetes cluster of Rational® Test Automation Server.
- If remote clusters are added to your project, the added clusters are displayed along with their availability status and ownership information.

- b. Select the location where you want to run the virtual service resource from the following options:

- Select the **Default Cluster** when no remote clusters are available in your project.
- Select the remote cluster from the list when a remote cluster is available in your project.
- Select **No override options**, if you selected any remote cluster and want to revert to the **Default Cluster** to run the virtual service resource.

- c. Click **Execute**.


The test run is initiated.

Results

You have started a virtual service from the **Resources** page on Rational® Test Automation Server.

What to do next

After you start a run of a virtual service, you can view the status of the virtual service in any of the following ways:

- On the **Resources** page, the virtual services are displayed with the state as `Running` along with the number of instances that are running in the **Active instances** column.
- On the **Resources** page, you must click the **Show in instances page** icon  in the row of the virtual service to view all the running instances of a particular virtual service on the **Instances** page.
- Go to the **Instances** page by clicking **Virtualization > Instances**. You can view instances of all the virtual services that are running and are displayed with the state as `Running` under the **State** column. You can also view the requests that are received by the virtual service. The number of requests is displayed as number of hits when you hover the cursor over the text in the **Activity** column.

You can view the details, configuration settings, routing rules, usage statistics, or logs of a specific running instance of the virtual service. See [Viewing configurations of running instances of virtual services on page 443](#).

You can modify the behavior or the logging level of a running instance of the virtual service that you started before running tests on them. See [Modifying configurations of running instances of virtual services on page 447](#).

When you have completed testing the virtual service, you must stop the running virtual service. See [Stopping virtual services on page 454](#).

Related reference

[Additional configuration parameters for virtual services on page 427](#)


Additional configuration parameters for virtual services



You can find information about the additional configuration parameters that you can set when you want to run virtual services on Rational® Test Automation Server.


When you use the server UI to start the virtual service, some of the parameters are handled by fields in the **Execute virtual service** dialog.


You can enter the other parameters as a *name-value* pair in the **Additional configuration parameters** field that is displayed in the **Advanced** settings panel of the **Execute virtual service** dialog.



The following additional configuration parameters are supported when you configure a run for a virtual service.

Requirement	Configuration parameter name	Supported values	An example value	Result of using the example value
To specify a unique identifier for use in the generated host name	<code>stub.ingress.host.identifier</code>	Any <code><custom_id></code> that you specify.  Note: The <code><custom_id></code> must start and end with	<code>mystub</code>	See Running HTTP virtual services without using proxies on page 433 , to know how the identifier you specified is used.

Requirement	Configuration parameter name	Supported values	An example value	Result of using the example value
		 an alphanumeric character. The characters supported include a-z, 0-9, and a hyphen. No other characters can be used in the <custom_id> . The <custom_id> must not be longer than 32 characters.		
To specify whether a virtual service instance should be run in a dedicated container.	<code>stub.dedicated.container</code>	<ul style="list-style-type: none"> • <i>true</i> • <i>false</i>  Note: The default value is <i>false</i> .	<i>true</i>	A dedicated container is used to run a single virtual service instance.
To specify whether performance optimization is enabled	<code>stub.performance.optimize</code>	<ul style="list-style-type: none"> • <i>true</i> • <i>false</i> 	<i>true</i>	The performance optimization is enabled for the virtual service.

Requirement	Configuration parameter name	Supported values	An example value	Result of using the example value
<p>for the virtual service instance. The value set overrides the setting from the stub definition.</p> <p>This option can be set in the Behavior tab from the UI.</p>		 Note: The default value is <i>false</i> .		
<p>To specify the number of threads to be used for processing requests for the virtual service instance. The value set overrides the setting from the stub definition.</p> <p>This option can be set in the Behavior tab from the UI.</p>	<code>stub.worker.thread-count</code>	Any number that you specify as number of threads to use.	15	15 threads are used for processing requests for the virtual service instance.
<p>To specify the stub logging level for the virtual service instance. The value set overrides the setting from the stub definition.</p> <p>This option can be set in the Advanced settings from the UI.</p>	<code>stub.logging.level</code>	<ul style="list-style-type: none"> • <i>none</i> • <i>normal</i> • <i>debug</i> 	<i>debug</i>	The virtual service writes informational and debugging messages to the container log.
<p>To specify the pass-through behaviour type of the virtual service instance. The value set overrides</p>	<code>stub.pass.through-behavior</code>	Valid values depend on the stub operation transport type but can be as follows:	<i>simulate_error</i>	This option returns an error to the calling system. The message is not passed to the system under test.

Requirement	Configuration parameter name	Supported values	An example value	Result of using the example value
<p>the setting from the stub definition.</p> <p>This option can be set in the Behavior tab from the UI.</p>		<ul style="list-style-type: none"> • <i>simulate_error</i> • <i>discard</i> • <i>pass_through</i> 		
<p>To specify the delay in passing through from the virtual service instance. The value set overrides the setting from the stub definition.</p>	<code>stub.pass.through.delay.period</code>	Any number of seconds in milliseconds.	<i>10000</i>	A delay of 10 seconds in passing through from the virtual service instance.
<p>To specify the status code to use in the virtual service instance response when simulating an error. The value set overrides the setting from the stub definition.</p> <p> Note: Applies to HTTP stubs.</p> <p>This option can be set in the Behavior tab from the UI.</p>	<code>stub.pass.through.status.code</code>	You must specify a valid HTTP status code as the value. For example, <i>503</i> .	<i>500</i>	The HTTP error code of <i>500</i> is sent in the response if the virtual service simulates an error when passing through.
<p>To specify the reason phrase to use in the virtual service instance response when simulating an error. The value set overrides the setting from the stub definition.</p>	<code>stub.pass.through.reason.phrase</code>	You must specify the text of the status message as the value.	<i>my custom error msg</i>	The text that you specify is set as the HTTP reason phrase in the response if the virtual service simulates an error when passing through.

Requirement	Configuration parameter name	Supported values	An example value	Result of using the example value
<p> Note: Applies to HTTP stubs.</p> <p>This option can be set in the Behavior tab from the UI.</p>				
<p>To specify the type of response-time behavior that is required for the virtual service instance. The value set overrides the setting from the stub definition.</p> <p>This option can be set in the Behavior tab from the UI.</p>	<pre>stub.response.time- .type</pre>	<ul style="list-style-type: none"> • <i>no_delay</i> • <i>minimum_delay</i> • <i>gaussian</i> • <i>uniform</i> • <i>performance_profile</i> <p> Note: <i>performance_profile</i> is only valid if the stub operation in the stub definition is defined to use a performance profile.</p>	<pre>no_delay</pre>	<p>The virtual service sends a response without any delay.</p>
<p>To specify the minimum time for the response time behaviour required for the virtual service instance. The value set overrides the setting from the stub definition.</p>	<pre>stub.response.time- .minimum</pre>	<p>Any number of seconds in milliseconds.</p>	<pre>5000</pre>	<p>The virtual service sends a response after a minimum delay of 5 seconds.</p>

Requirement	Configuration parameter name	Supported values	An example value	Result of using the example value
This option can be set in the Behavior tab from the UI.				
To specify the maximum time for the response time behaviour required for the virtual service instance. The value set overrides the setting from the stub definition. This option can be set in the Behavior tab from the UI.	<code>stub.response.time-maximum</code>	Any number of seconds in milliseconds.	<code>20000</code>	The virtual service must send a response after a maximum delay of 20 seconds.

Virtual services where the stub definition refers to multiple operations

When a stub definition has two or more events that refer to different operations and each operation has a different behavior, you must specify the `<operation_id>` suffixed to the configuration parameter name in the **Additional configuration parameters** field.



Note: The `<operation_id>` is the internal ID of the operation that is found in the **Documentation** tab when you open the operation in Rational® Integration Tester.

When the stub definition refers to multiple operations and you want to start a run of a virtual service, you must append the `<operation_id>` to the `<parameter_name>` in the **Additional configuration parameters** field.

The configuration parameter names that you must use for the virtual service, which is associated with a particular operation are as follows:

- `stub.pass.through.behavior.<operation_id>`
- `stub.pass.through.delay.period.<operation_id>`
- `stub.pass.through.status.code.<operation_id>`
- `stub.pass.through.reason.phrase.<operation_id>`
- `stub.response.time.type.<operation_id>`
- `stub.response.time.<operation_id>`
- `stub.response.time.maximum.<operation_id>`

For example, if the `<operation_id>` is `7281b750:162483a09f1:-7e68`, the parameter name for the `stub.response.time.type` option that you must specify is `stub.response.time.type.7281b750:162483a09f1:-7e68`.

Related information

[Configuring a run of a virtual service on page 419](#)

Running HTTP virtual services without using proxies

When you run HTTP virtual services on Rational® Test Automation Server, the endpoint on which the virtual service is exposed externally uses a host name that includes a generated ID. If a client application is required to call the virtual service directly via its external host name and not via the HTTP proxy, then you might want to determine the host name to configure the client first before you start the virtual service.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created a project in a team space on Rational® Test Automation Server and added the repository that contains the HTTP virtual services.
- Completed the prerequisite tasks before you start HTTP virtual services. See [Prerequisites for running HTTP virtual services on page 410](#).
- Selected the branch of the Git repository to view test resources on the **Execution** page.

About this task

When you installed Rational® Test Automation Server on Ubuntu, the default HTTP virtual service endpoints are of the following form:

```
in-<unique_id>.<ingress_domain>
```

For example, with an ingress domain as **10.1.2.3.nip.io**, a virtual service endpoint might be as follows:

```
in-cd3a755635574c6fa7e264911301821a.10.1.2.3.nip.io
```

When you installed Rational® Test Automation Server 10.2.1 or earlier on Red Hat OpenShift, the default HTTP virtual service endpoints are of the following form:

```
<ingress_domain_first_label>-<unique_id>.<remainder_of_ingress_domain>
```

For example, with an ingress domain as **rtas.mycluster-123456-0000.region.containers.appdomain.cloud**, a virtual service endpoint might be as follows:

```
rtas-cd3a755635574c6fa7e264911301821a.mycluster-123456-0000.region.containers.appdomain.cloud
```

The **<unique_id>** is generated when the virtual service is started. The full host name can be viewed in the **Routing to** field of the routing rule created for the virtual service on the **Intercept Rules** page. When traffic is routed to the virtual service via the HTTP proxy, a client application is unaware of this host name. If the client application is required to call the virtual services directly, it must be aware of the host name of the virtual service.

To allow a host name to be known before you start the virtual service, you can replace the generated ID with a user defined value that is specified as an additional configuration parameter when starting the stub. This value is referred to as **<custom_id>**.



Notes:

- The value of the **<custom_id>** that you specify is used as the **<unique_id>** for the virtual service.
- The **<custom_id>** must start and end with an alphanumeric character. The characters supported include a-z, 0-9, and a hyphen. No other characters can be used in the **<custom_id>**. The **<custom_id>** must not be longer than 32 characters.



Restriction: The **<custom_id>** must be unique to a virtual service that is running. You must not assign the same **<custom_id>** to another virtual service that is running simultaneously. Doing so might result in an undefined behavior of the virtual service.

For example, on Ubuntu, if you specify the value of the **<custom_id>** for the virtual service as `stub123-test` and Rational® Test Automation Server has an ingress domain as **10.1.2.3.nip.io**. Then the host name of the virtual service endpoint is as follows:

```
in-stub123-test.10.1.2.3.nip.io
```

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the virtual service resources in the test assets by clicking **Projects > My Projects > *project_name***.

The **Overview** page is displayed.



3. Click **Virtualization > Resources** in the navigation pane.

The **Resources** page is displayed.

4. Select the branch of the repository that contains the virtual services that you want to run from the list in the **Branch** field.

All virtual services in the selected branch are displayed on the **Resources** page.

5. Identify the virtual service that you want to run.

6. Click the **Open action menu** icon  in the row of the identified virtual service, and then click **Execute** icon .

The **Execute virtual service** dialog is displayed.

7. Select the version of the virtual service that is in the repository that you want to start by performing any of the following actions:
- Expand the list in the **Version** field, find the version of the test resources, and then select the version.

Use the following details about the version of the test resources that are displayed to identify the version that you want:


- Commit message.
- Tags labeled for the version committed.
- The user who committed the version to the repository.
- Relative time of the commit. For example, *2 hours ago* or *3 days ago*.

The list displays the versions of the test resources committed by all users to the branch in the repository. The versions are arranged with the latest version committed followed by the versions committed previously.

- Expand the list in the **Version** field, and then search for the version that you want to select by entering a partial or the complete commit message of that version.

The version that matches the search criteria is displayed and it is selected for the test run.

8. Select the environment that was used to bind the physical and logical resource in the API project, in the **ENVIRONMENT** tab.

 **Important:** The configuration that you set for the run in the **Execute virtual service** dialog is preserved when you run the same virtual service again. The configurations that you set are not available to other members when they want to run the virtual service. For example, if you selected an environment, the same environment is selected when you run the virtual service again.

9. Enter a label, if required.

A label that you enter for the test run that helps you to identify the virtual service instance on the **Instances** page. The label that you entered is displayed for the virtual service under the **Labels** column on the **Instances** page. After you have created a label, any member of the project can use that label.

10. Follow the instructions if you want to modify the configurations for the behavior of the virtual service:
- a. Click the **BEHAVIOR** tab, if it is not already open.
 - b. Configure or change the settings for the following options:



Note: The settings displayed are the settings that were configured for the virtual service when it was authored in Rational® Integration Tester.

Option	Description						
Performance	<p>The following settings are available for handling of requests by the virtual service:</p> <table border="1" data-bbox="574 533 1421 1654"> <thead> <tr> <th data-bbox="574 533 889 588">Option</th> <th data-bbox="889 533 1421 588">Description</th> </tr> </thead> <tbody> <tr> <td data-bbox="574 588 889 1482">Optimize performance</td> <td data-bbox="889 588 1421 1482"> <p>If enabled, attempts are made to reduce the amount of processing between the time the virtual service receives a request and the time it sends a response. Specific optimizations depend on the message contents, as in the following examples:</p> <ul style="list-style-type: none"> ▪ When the virtual service receives requests, all validations are disabled, and for all XML payloads, the store and filter actions are converted to use XPath expressions. ▪ When the virtualization sends responses, any store actions set on a message are disabled, and any XML content is collapsed when the virtual service is compiled instead of being collapsed every time that a response is sent. <p>The optimization is disabled as the default action for this setting and you can enable the performance optimization if you fully understand the implications of optimizing the performance.</p> </td> </tr> <tr> <td data-bbox="574 1482 889 1654">Threads</td> <td data-bbox="889 1482 1421 1654"> <p>The maximum number of threads used in processing requests received by the virtual service. The default number of threads is <i>10</i>.</p> </td> </tr> </tbody> </table>	Option	Description	Optimize performance	<p>If enabled, attempts are made to reduce the amount of processing between the time the virtual service receives a request and the time it sends a response. Specific optimizations depend on the message contents, as in the following examples:</p> <ul style="list-style-type: none"> ▪ When the virtual service receives requests, all validations are disabled, and for all XML payloads, the store and filter actions are converted to use XPath expressions. ▪ When the virtualization sends responses, any store actions set on a message are disabled, and any XML content is collapsed when the virtual service is compiled instead of being collapsed every time that a response is sent. <p>The optimization is disabled as the default action for this setting and you can enable the performance optimization if you fully understand the implications of optimizing the performance.</p>	Threads	<p>The maximum number of threads used in processing requests received by the virtual service. The default number of threads is <i>10</i>.</p>
Option	Description						
Optimize performance	<p>If enabled, attempts are made to reduce the amount of processing between the time the virtual service receives a request and the time it sends a response. Specific optimizations depend on the message contents, as in the following examples:</p> <ul style="list-style-type: none"> ▪ When the virtual service receives requests, all validations are disabled, and for all XML payloads, the store and filter actions are converted to use XPath expressions. ▪ When the virtualization sends responses, any store actions set on a message are disabled, and any XML content is collapsed when the virtual service is compiled instead of being collapsed every time that a response is sent. <p>The optimization is disabled as the default action for this setting and you can enable the performance optimization if you fully understand the implications of optimizing the performance.</p>						
Threads	<p>The maximum number of threads used in processing requests received by the virtual service. The default number of threads is <i>10</i>.</p>						
Operation	Specifies an operation referenced by the virtual service.						
Response time	Specifies the response time behavior for responses sent by the virtual service for the selected operation.						

Option	Description										
	<p>You can modify the value by selecting a value from the following options:</p> <table border="1" data-bbox="574 321 1419 949"> <thead> <tr> <th data-bbox="574 321 727 380">Option</th> <th data-bbox="727 321 1419 380">Description</th> </tr> </thead> <tbody> <tr> <td data-bbox="574 380 727 474">No delay</td> <td data-bbox="727 380 1419 474">Select this option for a response with no delay.</td> </tr> <tr> <td data-bbox="574 474 727 604">Minimum delay</td> <td data-bbox="727 474 1419 604">Select this option for a delay in the response by entering the required delay in milliseconds (ms).</td> </tr> <tr> <td data-bbox="574 604 727 779">Uniform distribution</td> <td data-bbox="727 604 1419 779">Select this option for a uniformly distributed response time by specifying the minimum and maximum delay in milliseconds (ms).</td> </tr> <tr> <td data-bbox="574 779 727 949">Gaussian distribution</td> <td data-bbox="727 779 1419 949">Select this option for a response time with Gaussian distribution by specifying the minimum and maximum delay in milliseconds (ms).</td> </tr> </tbody> </table>	Option	Description	No delay	Select this option for a response with no delay.	Minimum delay	Select this option for a delay in the response by entering the required delay in milliseconds (ms).	Uniform distribution	Select this option for a uniformly distributed response time by specifying the minimum and maximum delay in milliseconds (ms).	Gaussian distribution	Select this option for a response time with Gaussian distribution by specifying the minimum and maximum delay in milliseconds (ms).
Option	Description										
No delay	Select this option for a response with no delay.										
Minimum delay	Select this option for a delay in the response by entering the required delay in milliseconds (ms).										
Uniform distribution	Select this option for a uniformly distributed response time by specifying the minimum and maximum delay in milliseconds (ms).										
Gaussian distribution	Select this option for a response time with Gaussian distribution by specifying the minimum and maximum delay in milliseconds (ms).										
Passthrough	<p>Specifies the pass through behavior for the selected operation when requests are not handled within the virtual service.</p> <p>You can modify the value by selecting a value from the following options:</p> <table border="1" data-bbox="574 1161 1419 1491"> <tbody> <tr> <td data-bbox="574 1161 727 1304">Discard</td> <td data-bbox="727 1161 1419 1304">This option stops the system under test from receiving the intercepted message. This option can disrupt the calling system. For example, the system might time out while it waits for a reply.</td> </tr> <tr> <td data-bbox="574 1304 727 1398">Pass Through</td> <td data-bbox="727 1304 1419 1398">This option passes the intercepted message to the system under test, with an optional delay.</td> </tr> <tr> <td data-bbox="574 1398 727 1491">Simulate Error</td> <td data-bbox="727 1398 1419 1491">This option returns an error to the calling system. The message is not passed to the system under test.</td> </tr> </tbody> </table>	Discard	This option stops the system under test from receiving the intercepted message. This option can disrupt the calling system. For example, the system might time out while it waits for a reply.	Pass Through	This option passes the intercepted message to the system under test, with an optional delay.	Simulate Error	This option returns an error to the calling system. The message is not passed to the system under test.				
Discard	This option stops the system under test from receiving the intercepted message. This option can disrupt the calling system. For example, the system might time out while it waits for a reply.										
Pass Through	This option passes the intercepted message to the system under test, with an optional delay.										
Simulate Error	This option returns an error to the calling system. The message is not passed to the system under test.										

11. Click **Advanced** to make the following advanced configurations:

- a. Enter the following configuration options as parameters and their values in the **Additional Configuration Parameters** fields.

Parameter name	Value
stub.ingress.host.identifier	<custom_id>

12. Click **Execute**.


The run is initiated.

Results

You have started an HTTP virtual service that does not use a proxy. The virtual service exposes the host name that contains a custom ID that you specified.

What to do next

After you start a run of a virtual service, you can view the status of the virtual service in any of the following ways:

- On the **Resources** page, the virtual services are displayed with the state as `Running` along with the number of instances that are running in the **Active instances** column.
- On the **Resources** page, you must click the **Show in instances page** icon  in the row of the virtual service to view all the running instances of a particular virtual service on the **Instances** page.
- Go to the **Instances** page by clicking **Virtualization > Instances**. You can view instances of all the virtual services that are running and are displayed with the state as `Running` under the **State** column. You can also view the requests that are received by the virtual service. The number of requests is displayed as number of hits when you hover the cursor over the text in the **Activity** column.

Viewing running instances of virtual services

After you start virtual services from the **Resources** page, you can view the instances of the virtual services that are running from the **Instances** page in IBM® Rational® Test Automation Server. You can view the details, configuration settings, routing rules, usage statistics, or logs of a specific running instance of the virtual service. You can also stop the running instance.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Viewer*, *Tester*, or *Project Owner* in the project.
- Started the virtual service resources as a *Tester* or *Owner* from the **Resources** page.

About this task

After you started the virtual services from the **Resources** page on Rational® Test Automation Server, you can go to the **Instances** page and you can perform the following tasks:

- View all running instances of the virtual services that are contained in the project repositories.
- View the components in the system model that are associated with the virtual service instances.
- Search for a specific instance of the virtual service for which you want to view the details.

- Create or use saved filters for viewing specific running instances of the virtual service.
- View the details, configuration settings, routing rules, usage statistics, or logs of a specific running instance of the virtual service.
- Change the behavior or logging level of a running instance of the virtual service before you run tests on the virtual service.
- Stop a specific running instance of the virtual service.
- Stop multiple instances or all instances of the virtual service that are running.

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the virtual service resources in the test assets by clicking **Projects > My Projects > *project_name***.

The **Overview** page is displayed.

3. Click **Virtualization > Instances** in the navigation pane.

The **Instances** page is displayed.

Any one of the following views is displayed:

- No instances of virtual services are displayed if there are no instances that are running or were stopped in the past hour.
- All running instances of virtual services in your project are displayed.
- No running instances of virtual services are displayed but the option **Show all** is enabled. You can click **Show all** to display the instances that were stopped in the past hour. Instances that were hidden from the display when **Hide inactive** was clicked are displayed.



Note: All instances that were stopped are automatically removed from the display after *60 minutes* of being stopped. You cannot use the **Instances** page to view the history of stopped instances. You can hide the stopped instances from the display by clicking **Hide inactive**, so that you can view only the running instances.


You can find the following information about the running instances that are displayed on the **Instances** page.





- The name and path of the virtual service in the project repository are displayed in the **Name** column.
- The tags or labels added to the instance when the run was configured is displayed in the **Labels** column.
- The name of the environment associated with the operation in which the virtual service was created in Rational® Integration Tester is displayed in the **Environment** column.
- The state of the running instance is displayed as *Running* in the **State** column.
- The activity of the running instance of the virtual service based on the number of requests received in the previous hour is displayed in the **Activity** column. The activities are classified and displayed as indicated in the following table:





Activity load	Activity displayed as
Received 0 requests	None
Received 1 - 10 requests	Low
Received 11 - 100 requests	Mid
Received 101 - 1000 requests	High
Received more than 1000 requests	Very High

- The **Stop** icon  is displayed in the **Actions** column and stops the running instance of the virtual service, when clicked.

4. Complete the steps for the task that you want to perform as listed in the following table:

Task	Action
Searching for a specific instance of the virtual service for which you want to view the details or want to stop the instance.	<p>Enter either the complete text or part of the text in the name or the path of the virtual service that is in the repository in the Search field.</p> <p>The instances of the virtual services that match the text searched are displayed.</p>
Viewing the component associated with the running instance of the virtual service.	<p>You can view the name of the associated component that is displayed on the Instances page in the Filter row only if you clicked the icon  on the System Model page to view all running instances of the virtual services .</p>
Creating or using saved filters for viewing specific instances of virtual service resources.	<p>To create a query, perform the following actions:</p> <ol style="list-style-type: none"> Click New filter. Create a rule with an appropriate operator. Select criteria such as <i>Name, Path, Labels, Environment, State, or Activity</i>. Select the condition, and then enter the value for the criteria. Apply the filter query. <p>The virtual services that match the filter criteria are displayed.</p> <ol style="list-style-type: none"> Save the filter query for retrieving it from the saved filters list. <p>To use a saved filter query, perform the following actions:</p>

Task	Action
	<p> Note: To open the filter query, you must have created and saved a filter query.</p> <ol style="list-style-type: none"> Click the Open filters icon . Select the saved filter. Apply the filter. <p>The virtual services that match the filter criteria are displayed.</p>
Viewing the details, configuration settings, routing rules, usage statistics, or logs of a specific running instance of the virtual service.	<p>After you identify the virtual service instance for which you want to view the details, perform the following actions:</p> <ol style="list-style-type: none"> Click the Expand icon  in the row of the identified instance of the virtual service. <p>The Details panel of the instance is displayed. You can find the following details about the instance displayed:</p> <ul style="list-style-type: none"> ▪ Name ▪ Path ▪ Labels ▪ API Environment ▪ Version ▪ Started at ▪ Started by ▪ Associated components ▪ Documentation <ol style="list-style-type: none"> Click Behavior to expand the panel and view the behavior that was configured for the virtual service. Click Routing Rules to expand the panel and view the routing rules that were configured in the virtual service. Click Statistics to expand the panel and view the graphical representation of the activities performed by the virtual service. Click Diagnostics to expand the panel and view the logging configuration of the virtual service and the links to the logs.
Changing the behavior or logging level of a running instance of the virtual service.	<p>After you identify the virtual service instance for which you want to view the details, perform the following actions:</p> <ol style="list-style-type: none"> Click the Expand icon  in the row of the identified instance of the virtual service. <p>The Details panel of the instance is displayed.</p>

Task	Action
	<p>Perform any of the following actions:</p> <ul style="list-style-type: none"> ◦ Click Behavior to expand the panel. <p>Perform the following steps to change the behavior configurations:</p> <ol style="list-style-type: none"> a. Click the Edit icon . b. Select a different operation for the virtual service, if available. c. Select a different response time and enter the values for the delay that you want to set. d. Click apply. <ul style="list-style-type: none"> ◦ Click Diagnostics to expand the panel. <p>Perform the following steps to change the logging level configuration:</p> <ol style="list-style-type: none"> a. Click the Edit icon . b. Select a different logging level. c. Click apply.
Stopping a specific running instance of the virtual service.	<ol style="list-style-type: none"> a. Click the Stop icon  in the row of the virtual service instance. b. Click Ok in the Stop virtual service instance dialog that is displayed. <p>The running virtual service instance is stopped.</p>
Stopping multiple running instances or all the running instances of virtual services.	<ol style="list-style-type: none"> a. Perform any of the following actions: <ul style="list-style-type: none"> ▪ Select the checkbox in the row of the virtual service instances that you want to stop, and then click Stop selected. ▪ Select the Select all checkbox in the header row, and then click Stop selected. <p> Note: All running instances are selected when you click the Select all checkbox.</p> b. Click Ok in the Stop all selected virtual service instances dialog that is displayed. <p>The running virtual service instances are stopped.</p>

Results

You have viewed running instances of the virtual services from the **Instances** page on Rational® Test Automation Server.

What to do next

You can perform the following tasks:

- View the configurations of a running instance of the virtual service. See [Viewing configurations of running instances of virtual services on page 443](#).
- Modify the behavior or logging level of a running instance of the virtual service before you run tests on the virtual service. See [Modifying configurations of running instances of virtual services on page 447](#).
- Stop a running instance or all running instances of the virtual services. See [Stopping virtual services on page 454](#).

Viewing configurations of running instances of virtual services

After you start virtual services from the **Resources** page, you can view the details, configuration settings, routing rules, usage statistics, or logs of a specific running virtual service instance from the **Instances** page on IBM® Rational® Test Automation Server.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Viewer*, *Tester*, or *Project Owner* in the project.
- Started the virtual service resources as a *Tester* or *Owner* from the **Resources** page.

About this task

After you have started the virtual services from the **Resources** page on Rational® Test Automation Server, you can go to the **Instances** page and view the information about a specific running virtual service instance.

1. Log in to Rational® Test Automation Server.


The team space that contains your project is displayed.

2. Open the project that contains the virtual service resources in the test assets by clicking **Projects > My Projects > *project_name***.

The **Overview** page is displayed.

3. Click **Virtualization > Instances** in the navigation pane.

The **Instances** page is displayed with all running instances of the virtual services that are in your project.

4. Identify the running instance of the virtual service of which you want to view the details, configuration settings, and other information.
5. Click the **Expand** icon  of the instance of the virtual service.

The **Details** panel of the instance is displayed with the following details about the virtual service instance:





Note: The information displayed pertains to the configuration settings that were configured in the virtual service at the time it was created or run.














Option	Information displayed
Name	Displays the name of the virtual service.
Path	Displays the path of the virtual service resource in the project repository.
Labels	Displays the labels or tags that were entered for the virtual service when a run of the virtual service was configured.
API Environment	Displays the name of the environment associated with the operation in which the virtual service was created in Rational® Integration Tester or the environment selected for the run.
Version	Displays the version of the virtual service resource that is in the project repository that was started as an instance.
Started at	Displays the date and time when the virtual service instance started.
Started by	Displays the name of the project member who started the virtual service instance.
Associated components	Displays the components in the system model that are associated with the virtual service.
Documentation	Displays the text that was entered on the Documentation tab of the Stub Editor when the virtual service was created in Rational® Integration Tester.


The other panels that you can view are as follows:

- Behavior
 - Routing Rules
 - Statistics
 - Diagnostics
6. Expand each panel by clicking the panel to view the information in that panel.

You can view the following information in the different panels:

Panel	Details										
Behavior	<p>The Behavior panel of the instance is displayed when expanded with the following details about the virtual service instance:</p> <p> Note: The information displayed pertains to the configuration settings that were configured in the virtual service at the time it was created or run.</p> <table border="1" data-bbox="524 533 1419 1054"> <thead> <tr> <th data-bbox="524 533 703 590">Option</th> <th data-bbox="703 533 1419 590">Information displayed</th> </tr> </thead> <tbody> <tr> <td data-bbox="524 590 703 688">Performance</td> <td data-bbox="703 590 1419 688">Displays the optimization setting and the maximum number of threads in use by the virtual service.</td> </tr> <tr> <td data-bbox="524 688 703 827">Operation</td> <td data-bbox="703 688 1419 827">Displays the operations referenced by the virtual service. You can select each operation to view the related response time and the pass through settings.</td> </tr> <tr> <td data-bbox="524 827 703 926">Response time</td> <td data-bbox="703 827 1419 926">Displays the response time option that is in use for the selected operation within the virtual service.</td> </tr> <tr> <td data-bbox="524 926 703 1054">Passthrough</td> <td data-bbox="703 926 1419 1054">Displays the pass through option that is in use for the selected operation within the virtual service.</td> </tr> </tbody> </table>	Option	Information displayed	Performance	Displays the optimization setting and the maximum number of threads in use by the virtual service.	Operation	Displays the operations referenced by the virtual service. You can select each operation to view the related response time and the pass through settings.	Response time	Displays the response time option that is in use for the selected operation within the virtual service.	Passthrough	Displays the pass through option that is in use for the selected operation within the virtual service.
Option	Information displayed										
Performance	Displays the optimization setting and the maximum number of threads in use by the virtual service.										
Operation	Displays the operations referenced by the virtual service. You can select each operation to view the related response time and the pass through settings.										
Response time	Displays the response time option that is in use for the selected operation within the virtual service.										
Passthrough	Displays the pass through option that is in use for the selected operation within the virtual service.										
Routing Rules	<p>The Routing Rules panel of the instance is displayed when expanded.</p> <p>The routing rules that relate to the virtual service are displayed.</p> <table border="1" data-bbox="524 1230 1419 1646"> <thead> <tr> <th data-bbox="524 1230 654 1287">Column</th> <th data-bbox="654 1230 1419 1287">Description</th> </tr> </thead> <tbody> <tr> <td data-bbox="524 1287 654 1381">Activity</td> <td data-bbox="654 1287 1419 1381">Indicates the routing type.</td> </tr> <tr> <td data-bbox="524 1381 654 1514">Target</td> <td data-bbox="654 1381 1419 1514">Displays the endpoint of the target service for which requests are being routed.</td> </tr> <tr> <td data-bbox="524 1514 654 1646">Recipient</td> <td data-bbox="654 1514 1419 1646">Displays the endpoint of the virtual service that receives the routed requests</td> </tr> </tbody> </table> <p>You can expand the panel for each intercept by clicking on the expand icon . You can then find the following information:</p>	Column	Description	Activity	Indicates the routing type.	Target	Displays the endpoint of the target service for which requests are being routed.	Recipient	Displays the endpoint of the virtual service that receives the routed requests		
Column	Description										
Activity	Indicates the routing type.										
Target	Displays the endpoint of the target service for which requests are being routed.										
Recipient	Displays the endpoint of the virtual service that receives the routed requests										

Panel	Details																	
	<table border="1"> <thead> <tr> <th data-bbox="522 260 656 310">Tab</th> <th data-bbox="656 260 1414 310">Description</th> </tr> </thead> <tbody> <tr> <td data-bbox="522 310 656 1192">Details</td> <td data-bbox="656 310 1414 1192"> <p>The Details panel provides the following information:</p> <table border="1"> <thead> <tr> <th data-bbox="667 386 789 441">Option</th> <th data-bbox="789 386 1403 441">Information displayed</th> </tr> </thead> <tbody> <tr> <td data-bbox="667 441 789 890">Condition</td> <td data-bbox="789 441 1403 890"> <p>Lists the condition that causes the traffic to be routed to a virtual service.</p> <p>You can view the detailed condition of the rule defined for the intercept by clicking the View icon .</p> <p>The detailed condition of the rule is displayed in the Full condition window.</p> <p> Note: You can copy the detailed condition if you intend to use it elsewhere.</p> </td> </tr> <tr> <td data-bbox="667 890 789 989">Routing to</td> <td data-bbox="789 890 1403 989">Lists the virtual service to which traffic is routed.</td> </tr> <tr> <td data-bbox="667 989 789 1087">Created by</td> <td data-bbox="789 989 1403 1087">Lists the user who created the rule.</td> </tr> <tr> <td data-bbox="667 1087 789 1186">Created at</td> <td data-bbox="789 1087 1403 1186">Lists the date and time when the rule was created.</td> </tr> </tbody> </table> </td> </tr> <tr> <td data-bbox="522 1192 656 1333">Activity</td> <td data-bbox="656 1192 1414 1333"> <p>The Activity panel lists the log of the activities performed in relation to the routing rule. This panel also displays the details of the activity, date and time, and message for that activity.</p> </td> </tr> </tbody> </table>	Tab	Description	Details	<p>The Details panel provides the following information:</p> <table border="1"> <thead> <tr> <th data-bbox="667 386 789 441">Option</th> <th data-bbox="789 386 1403 441">Information displayed</th> </tr> </thead> <tbody> <tr> <td data-bbox="667 441 789 890">Condition</td> <td data-bbox="789 441 1403 890"> <p>Lists the condition that causes the traffic to be routed to a virtual service.</p> <p>You can view the detailed condition of the rule defined for the intercept by clicking the View icon .</p> <p>The detailed condition of the rule is displayed in the Full condition window.</p> <p> Note: You can copy the detailed condition if you intend to use it elsewhere.</p> </td> </tr> <tr> <td data-bbox="667 890 789 989">Routing to</td> <td data-bbox="789 890 1403 989">Lists the virtual service to which traffic is routed.</td> </tr> <tr> <td data-bbox="667 989 789 1087">Created by</td> <td data-bbox="789 989 1403 1087">Lists the user who created the rule.</td> </tr> <tr> <td data-bbox="667 1087 789 1186">Created at</td> <td data-bbox="789 1087 1403 1186">Lists the date and time when the rule was created.</td> </tr> </tbody> </table>	Option	Information displayed	Condition	<p>Lists the condition that causes the traffic to be routed to a virtual service.</p> <p>You can view the detailed condition of the rule defined for the intercept by clicking the View icon .</p> <p>The detailed condition of the rule is displayed in the Full condition window.</p> <p> Note: You can copy the detailed condition if you intend to use it elsewhere.</p>	Routing to	Lists the virtual service to which traffic is routed.	Created by	Lists the user who created the rule.	Created at	Lists the date and time when the rule was created.	Activity	<p>The Activity panel lists the log of the activities performed in relation to the routing rule. This panel also displays the details of the activity, date and time, and message for that activity.</p>	
Tab	Description																	
Details	<p>The Details panel provides the following information:</p> <table border="1"> <thead> <tr> <th data-bbox="667 386 789 441">Option</th> <th data-bbox="789 386 1403 441">Information displayed</th> </tr> </thead> <tbody> <tr> <td data-bbox="667 441 789 890">Condition</td> <td data-bbox="789 441 1403 890"> <p>Lists the condition that causes the traffic to be routed to a virtual service.</p> <p>You can view the detailed condition of the rule defined for the intercept by clicking the View icon .</p> <p>The detailed condition of the rule is displayed in the Full condition window.</p> <p> Note: You can copy the detailed condition if you intend to use it elsewhere.</p> </td> </tr> <tr> <td data-bbox="667 890 789 989">Routing to</td> <td data-bbox="789 890 1403 989">Lists the virtual service to which traffic is routed.</td> </tr> <tr> <td data-bbox="667 989 789 1087">Created by</td> <td data-bbox="789 989 1403 1087">Lists the user who created the rule.</td> </tr> <tr> <td data-bbox="667 1087 789 1186">Created at</td> <td data-bbox="789 1087 1403 1186">Lists the date and time when the rule was created.</td> </tr> </tbody> </table>	Option	Information displayed	Condition	<p>Lists the condition that causes the traffic to be routed to a virtual service.</p> <p>You can view the detailed condition of the rule defined for the intercept by clicking the View icon .</p> <p>The detailed condition of the rule is displayed in the Full condition window.</p> <p> Note: You can copy the detailed condition if you intend to use it elsewhere.</p>	Routing to	Lists the virtual service to which traffic is routed.	Created by	Lists the user who created the rule.	Created at	Lists the date and time when the rule was created.							
Option	Information displayed																	
Condition	<p>Lists the condition that causes the traffic to be routed to a virtual service.</p> <p>You can view the detailed condition of the rule defined for the intercept by clicking the View icon .</p> <p>The detailed condition of the rule is displayed in the Full condition window.</p> <p> Note: You can copy the detailed condition if you intend to use it elsewhere.</p>																	
Routing to	Lists the virtual service to which traffic is routed.																	
Created by	Lists the user who created the rule.																	
Created at	Lists the date and time when the rule was created.																	
Activity	<p>The Activity panel lists the log of the activities performed in relation to the routing rule. This panel also displays the details of the activity, date and time, and message for that activity.</p>																	
Statistics	<p>The Statistics panel of the instance is displayed when expanded.</p> <p>The requests received by the instance of the virtual service in the previous hour are displayed as a graph. The total requests received is indicated in the Total hits field and the activity load in the previous hour are displayed in the Activity field.</p> <p>The view can be refreshed by clicking the Refresh icon .</p>																	
Diagnostics	<p>The Diagnostics panel of the instance is displayed when expanded with the following details:</p>																	

Panel	Details	
	Option	Description
	Logging	Displays the logging level that was configured for the virtual service.
	Init container log	Displays the link to the log of the Init container.  Note: This logs contains messages that relate to configuring the container which is ready to run the virtual service.
	Main container log	Displays the link to the log of the main container. This log contains messages logged by the virtual service based on the selected logging level.

Results

You have viewed the information about the configuration, routing rules, usage, or logs of the virtual service instance.

What to do next

You can modify the behavior or logging level configurations of the running virtual service instance before you run tests on the instance. See [Modifying configurations of running instances of virtual services on page 447](#).

Related information

[Configuring a run of a virtual service on page 419](#)

Modifying configurations of running instances of virtual services

After you start virtual services from the **Resources** page and before you run tests on the virtual service, you might want to modify the configurations for the behavior or the logging level of the running instance.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Viewer*, *Tester*, or *Project Owner* in the project.
- Started the virtual service resources as a *Tester* or *Owner* from the **Resources** page.

About this task

After you have started the virtual services from the **Resources** page on IBM® Rational® Test Automation Server, you can go to the **Instances** page and identify the running instance of the virtual service for which you want to modify the configurations. You can modify the configurations for the behavior or logging levels of the running instance of the virtual service.

1. Log in to Rational® Test Automation Server.


The team space that contains your project is displayed.

2. Open the project that contains the virtual service resources in the test assets by clicking **Projects > My Projects > project_name**.

The **Overview** page is displayed.

3. Click **Virtualization > Instances** in the navigation pane.

The **Instances** page is displayed with all running instances of the virtual services that are in your project.

4. Identify the running instance of the virtual service for which you want to modify the configurations.
5. Click the **Expand** icon  of the instance that you want to modify the configurations.

The **Details** panel of the instance is displayed.

6. Click **Behavior** to expand the panel and perform the following steps to change the configurations:

- a. Click the **Edit** icon .

The options that you can modify are enabled for editing.

- b. Select a different operation for the virtual service, if available, from the list for the **Operation** option.
- c. Select a different response time from the list for the **Response time** option, and then enter the values for the delay that you want to set.

You can modify the value by selecting a value from the following options:

Option	Description
No delay	Select this option for a response with no delay.
Minimum delay	Select this option for a delay in the response by entering the required delay in milliseconds (ms).
Uniform distribution	Select this option for a uniformly distributed response time by specifying the minimum and maximum delay in milliseconds (ms).

Option	Description
Gaussian distribution	Select this option for a response time with Gaussian distribution by specifying the minimum and maximum delay in milliseconds (ms).

- d. Click **apply** to apply the modified settings.
7. Click **Diagnostics** to expand the panel and perform the following steps to change the logging level that is configured:

- a. Click the **Edit** icon .

The **Logging level** option is enabled for editing.

- b. Select a different logging level from the following options.

Option	Description
None	Specifies that the virtual service does not write log messages.
Normal	Specifies that the virtual service writes informational messages.
Debug	Specifies that the virtual service writes informational and debugging messages.

- c. Click **apply** to apply the modified settings.

Results

You have viewed and modified the configurations of a running virtual service instance from the **Instances** page on Rational® Test Automation Server.

What to do next

You can run tests that are in an API Suite in your project, which then run on the virtual service instance. See [Configuring an API Suite or an API test run on page 267](#).

Viewing routing rules of the virtual services

When you run virtual services that are created to run on proxies or intercepts and have routing rules defined, you can view the proxies or intercepts and the routing rules from the **Intercept Rules** page.

Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Viewer*, *Tester*, or *Project Owner* in the project.
- Started the virtual service resources as a *Tester* or *Owner* from the **Resources** page.

About this task

After you run the virtual service resources that are in your project on Rational® Test Automation Server, you can view the routing rules for the running instances of the virtual service resources, from the **Intercept Rules** page.

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the virtual service resources in the test assets by clicking **Projects > My Projects > project_name**.

The **Overview** page is displayed.


3. Click **Virtualization > Intercept Rules** in the navigation pane.

The **Intercept Rules** page is displayed.

The agents, proxies, or intercepts that are configured in the virtual service resources that are in your project are displayed.

You can find the following information that is displayed on the **Intercept Rules** page.



Column	Description
Activity	Lists the agent or proxy.
Target	Lists a summary view of the endpoint that was recorded or virtualized.
Recipient	Lists the destination where messages sent to are captured.

4. Expand the proxy or intercept panel by clicking the **Expand** icon .

The details of the proxy or intercept are displayed in the **Details** tab in the expanded panel.

5. Click each tab to view the following information:

Tab	Description				
Details	The Details panel provides the following information:				
	<table border="1"> <thead> <tr> <th>Option</th> <th>Description</th> </tr> </thead> <tbody> <tr> <td>Condition</td> <td>Lists the condition that causes the traffic to be recorded or routed to a virtual service.</td> </tr> </tbody> </table>	Option	Description	Condition	Lists the condition that causes the traffic to be recorded or routed to a virtual service.
	Option	Description			
Condition	Lists the condition that causes the traffic to be recorded or routed to a virtual service.				

Tab	Description	
	Option	Description
		<p>You can view the detailed condition of the rule defined for the proxy or intercept by clicking the View icon .</p> <p>The detailed condition of the rule is displayed in the Full condition window.</p> <p> Note: You can copy the detailed condition if you intend to use it elsewhere.</p>
	Routing to	Lists the virtual service to which traffic is routed.
	Created by	Lists the user who created the rule.
	Created at	Lists the date and time when the rule was created.
Activity	The Activity panel lists the log of the activities performed by the proxy or intercept. This panel also displays the details of the activity, date and time, and message for that activity.	

Results

You have viewed the routing rules for the virtual service, proxy, or intercept that are running or connected to Rational® Test Automation Server.

What to do next

You can perform any of the following tasks:

- You can view the usage statistics of the virtual services that are running. See [Viewing usage statistics of virtual services on page 451](#).
- You can stop the running instances of the virtual services. See [Stopping virtual services on page 454](#).

Related information

[Management of virtualized services on page 408](#)

Viewing usage statistics of virtual services

After you start virtual service resources that are in the project repositories, and then run tests on the virtual service, you can view the usage statistics of the virtual services from the **Usage** page on IBM® Rational® Test Automation Server.

Before you begin

You must have completed the following tasks:

- Completed the prerequisite tasks before you configure a run of the HTTP virtual services. See [Prerequisites for running HTTP virtual services on page 410](#).
- Read the prerequisite information about virtualizing the Istio based services. See [Prerequisites for running virtualized Istio based services in IBM Rational Test Automation Server on page 412](#).
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Viewer*, *Tester*, or *Project Owner* in the project.
- Started the virtual service resources as a *Tester* or *Owner* from the **Resources** page.
- Ran API Suites with tests that run on the virtual service resources that are configured.

About this task

After you run the virtual services that are in a project, you can view details of the usage statistics of the virtual services from the **Usage** page on Rational® Test Automation Server.



Important: The settings such as the **Period**, or **From** and **To** that you configure to view the usage details are retained for you during your ongoing session and the settings are retained even when you perform the following activities:

- Log out of the server and log in again.
- Log in after a server restart.

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Open the project that contains the virtual service resources in the test assets by clicking **Projects > My Projects > *project_name***.

The **Overview** page is displayed.

3. Click **Virtualization > Usage** in the navigation pane.

The **Virtual Service Usage** page is displayed.


The total requests that were received by all virtual services on the server are displayed as a graph.

The default view displays the usage data of the virtual services that are gathered during the previous week and grouped in an hourly interval. The total requests received from when the virtual service was started are displayed.

The metric data that is reported is for the number of requests that are received by a virtual service. The requests can be requests, messages, or other calls that are received by the virtual service.

4. Change the default settings to view the graph for the period for which you want to view the usage.

You can change the default view by selecting an option from the following options:

Option	Description
<i>1 Hour</i>	Displays the usage data of the virtual service that are gathered during the previous hour and grouped in a 20-second interval.
<i>1 Day</i>	Displays the usage data of the virtual service that are gathered during the previous day and grouped in an eight-minute interval.
<i>1 Week</i>	Displays the usage data of the virtual service that are gathered during the previous week and grouped in an hourly interval.  Note: This is the default view.
<i>Range</i>	Displays the usage data of the virtual service that are gathered during the date range specified in the From and To fields. You can select the date and time for both the From and To fields. The data is dynamically grouped in intervals that depend on the range selected.



Note: You can refresh the view at any time to present the graph for the latest data by clicking **Refresh**.

- View details of the virtual service by hovering the mouse over any of the data-points to view the details at that point.

The following details about the virtual service are displayed when you hover the mouse over the data-point:

- The date and time of the request received.
- The number of requests received.

Results

You have viewed the usage details of the virtual services in your project that ran on Rational® Test Automation Server.

What to do next

You can perform any of the following tasks:

- You can view the routing rules defined in the virtual service, agent, proxy, or intercept. See [Viewing routing rules of the virtual services on page 449](#).
- You can stop the virtual service instances that are running. See [Stopping virtual services on page 454](#).

Related information

[Management of virtualized services on page 408](#)

Stopping virtual services

After you started instances of the virtual services from the **Resources** page, you must stop the running instance from the **Instances** page when you do not want to use the instance. You can stop a single instance, multiple, or all running instances.

Before you begin

You must have started the virtual service from the **Resources** page. See [Configuring a run of a virtual service on page 419](#).

1. Log in to Rational® Test Automation Server.


The team space that contains your project is displayed.

2. Open the project that contains the virtual service resources in the test assets by clicking **Projects > My Projects > project_name**.


The **Overview** page is displayed.


3. Go to the **Instances** page and identify the instance of the virtual service that you want to stop.

You can select the virtual service instances that satisfy the following conditions:

- The virtual service instance must be either in the **In transition Or Running state**.
- The **Stop** icon  in the **Actions** column must be enabled for the virtual service instance.
- The checkbox icon must be enabled for selection for the virtual service instance.

4. Perform the actions indicated in the following table:

If...	Action...
You want to stop any virtual service instance that is running	Perform any of the following actions: <ul style="list-style-type: none"> ◦ Click the Stop icon  in the Actions column of the virtual service instance. The Stop virtual service instance dialog is displayed. ◦ Select the checkbox in the row of the virtual service instance, and then click Stop selected. The Stop all selected virtual service instances dialog is displayed.

If...	Action...
You want to stop multiple virtual service instances that are running	Perform any of the following actions: <ul style="list-style-type: none"> ◦ Select the checkbox in the row of the virtual service instance, and then click Stop selected. ◦ Select the Select all checkbox in the header row, and then click Stop selected. <div style="text-align: center;">  Note: All running instances are selected when you select the Select all checkbox. </div> <p>The Stop all selected virtual service instances dialog is displayed.</p>

5. Click **Ok**.

A notification is displayed that the running virtual service instances are stopped.

Results

You have stopped a single instance, multiple, or all running virtual service instances. The stopped virtual service instance on the **Instances** page is displayed with the status as *Stopped by User* in the **State** column.

What to do next

You can restart the virtual service by completing the configurations that you want from the **Resources** page.

You can open and view the `Main container log` of the stopped virtual service.

Test results

After the tests, Suites, or schedules are run and completed, you can view the results and open the reports in IBM® Rational® Test Automation Server to analyze the verdict, the performance, and statistics. You can also re-execute tests and schedules from the **Result** page with the same commit id.

Summary

The **Results** page displays the verdict summary for all the tests run in the project.

In this page, you can perform any of the following tasks:

- Display the suites only, or the tests only, or tests and suites that are run with advanced settings.
- Searching for test results. You can filter test results by using any of the following ways:
 - By using the **Search** field to search for results by name.
 - By selecting predefined time interval from the **Selection interval** drop-down list.
 - By selecting **To** and **From date** from the **Date Interval** option.

- By clicking verdict from the **Verdict summary** slider (*Pass, Fail, Inconclusive, or Error*).
- By creating filter queries.
- Adding labels if you have a tester or an owner role.
- Adding text in the **Notes** field. You can also group the notes similar to a conversation in a chat channel.
- Locking test results: Locked results can be unlocked by the project owner or the project member who locked the results.
- Deleting test results.
- Comparing performance reports.
- Viewing trending reports.
- Viewing multiple reports depending on the test types and the parameters set when you execute the tests:
 - Statistics reports, Functional Test reports, Test logs or Result data queries
- Re-executing tests with the same commit id.

For more details, see the links in the next section of this page.

Test results and reports

Rational® Test Automation Server is a single location for hosting the results and reports of all tests run on different desktop clients and for tests run from the server.

Test results list

When a test run is complete, you can see the results on the **Results** page that you open from **Analyze > Results** in the navigation pane.

The list of test results is organized by types, you can display the suites only in the **Suites** tab, or the tests only in the **Tests** tab, or the tests and suites that are run with advanced settings in the **Advanced** tab.

The screenshot shows the 'TESTS' tab selected. At the top, there are three tabs: 'SUITES', 'TESTS', and 'ADVANCED'. Below the tabs is a 'Verdict summary' bar with a green segment for 'Pass 63.6%', a red segment for 'Fail 9.1%', and an orange segment for 'Error 27.3%'. Below the bar is a search bar and a table of test results.

Type	Test	Started by	Started on	Duration	Components	Labels	Actions
<input type="checkbox"/>	draco remoteagentproj/draco.testsuite	nm	2 days ago	00:01:48.037			+ [lock] [menu]
<input type="checkbox"/>	Simple test for addition Sample Integration Project/Logi...	nm	23 days ago	00:00:15.085			+ [lock] [menu]

At the bottom of the table, there is a pagination bar showing page 1 of 15.


When you expand the results, the results details and reports cards are displayed.

Results, reports, and logs are generated for tests run from Rational® Test Automation Server, or from desktop clients such as Rational® Performance Tester, Rational® Functional Tester, or Rational® Integration Tester.

You must have configured the desktop clients to publish reports of tests that are run from the desktop client to the Rational® Test Automation Server. For more information about the publishing procedure, refer to the links at the end of this page.

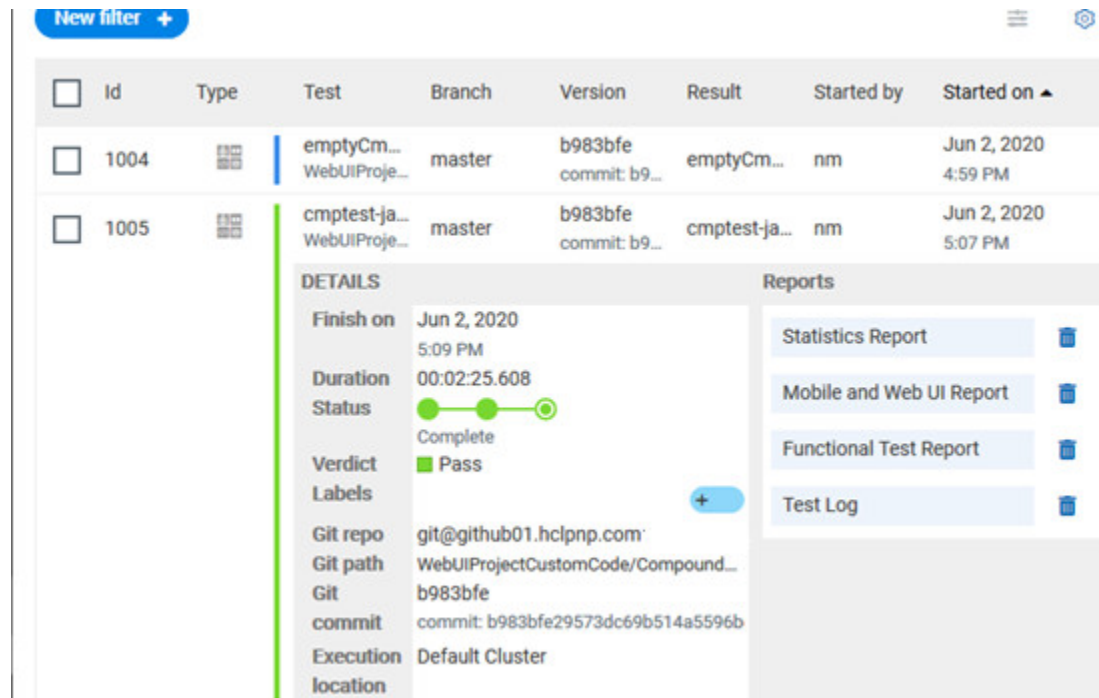
Test result details

By using the default settings, you can view the following details about the test assets used in the test run in the Details card:



You can configure the information displayed in the **Details** card and the **Results** columns from the **View Settings** window. You access this window by clicking the **Settings** icon .

Depending on the attribute you cleared in the **View Settings** window, the **Details** card displays additional information about the test results.




The selected attributes are displayed in columns of the Result view.







The screenshot shows a table of test results with columns: Id, Type, Test, Branch, Version, Result, Started by, and Started on. Two test runs are listed: ID 1004 and ID 1005. Below the table, the details for test ID 1005 are expanded, showing a 'DETAILS' card and a 'Reports' card.

Id	Type	Test	Branch	Version	Result	Started by	Started on
1004		emptyCm... WebUIProje...	master	b983bfe commit: b9...	emptyCm...	nm	Jun 2, 2020 4:59 PM
1005		cmptest-ja... WebUIProje...	master	b983bfe commit: b9...	cmptest-ja...	nm	Jun 2, 2020 5:07 PM

DETAILS

- Finish on: Jun 2, 2020 5:09 PM
- Duration: 00:02:25.608
- Status:  Complete
- Verdict:  Pass
- Labels: 
- Git repo: git@github01.hclpnp.com:
- Git path: WebUIProjectCustomCode/Compound...
- Git commit: b983bfe
- Execution location: Default Cluster

Reports

- Statistics Report 
- Mobile and Web UI Report 
- Functional Test Report 
- Test Log 

Reports

The Reports card contains the links to the test reports and the logs that are displayed in a web browser.

As a default configuration, test logs are delivered in a traditional format for the executed compound tests and schedules. You can still set the `-history jaeger` Program Argument to produce Jaeger traces when you run the tests.

The reports are generated for Compound Tests, Schedules, and Test Suites. They can be run from a desktop client or from Rational® Test Automation Server.

The following table lists the reports that are generated for each type of test and the links to help pages.

Table 10. Analyzing Reports

Report	Tests type	Product	More information
Functional Test Report	API Suite	IBM® Rational® Integration Tester	Viewing reports
	Compound Test	IBM® Rational® Functional Tester	Unified Report
	AFT Suite		
	Functional test (For Web UI tests)		
Mobile and Web UI Report	Compound Test	IBM® Rational® Functional Tester and IBM® Rational® Performance Tester	Mobile and web UI test result reports
	AFT Suite	IBM® Rational® Functional Tester	
Statistics Report	Compound Test	IBM® Rational® Functional Tester and IBM® Rational® Performance Tester	Reports and counters . Service Performance report . From the Statistic reports, you can add counters. Refer to Adding additional counters on a separate page.. You can also display the statistics of counters on graphs. Refer to Displaying counter data in tables or as graphs. .
	AFT Suite	IBM® Rational® Functional Tester	
	VU Schedule	IBM® Rational® Performance Tester	
	Rate Schedule		
	Functional test (For Web UI tests)		
Test Log	Compound Test	IBM® Rational® Functional Tester and IBM® Rational® Performance Tester	
	AFT Suite	IBM® Rational® Functional Tester	
	API Suite	IBM® Rational® Integration Tester	

Table 10. Analyzing Reports (continued)

Report	Tests type	Product	More information
	Functional Test (Web UI)	IBM® Rational® Functional Tester	
	VU Schedule	IBM® Rational® Performance Tester	
	Rate Schedule		

Creating search queries to filter test results

To filter the test and suite results list, you can create a search query that you apply as a filter in IBM® Rational® Test Automation Server.

About this task

To filter the list of test and suite results, you can use rules and group of rules in your query. This procedure explains how to create a query that contains two rules and a group of rules as an example, and how to select other filters.

1. Click **New filter** on the Result page.

Follow these steps to add a rule in your query:

2. Select **Add a rule** in your query, and enter or select the required attribute in the fields to define the rule:

- a. Select the first attribute that you want to query in the drop-down list.

Example

Select **Verdict**.

- b. Select the condition as *equal to (=)* or *not equal to (!=)*, *Contain* or *Not contain*.

Example

Select **=**.

- c. Select a value for the first attribute you selected.

The content of this field depends on the option you selected as first attribute. Enter a value for the **id**, **TypeTest**, **Version**, **Result**, **Labels**, **Branch**, **Test path**, and **Text** attributes. For the other attributes, select a value in the drop-down list.

Example

Select **Fail**.

- d. Select either the **AND** or the **OR** operator in the first field to add a condition to compare the first rule with another rule.

- e. Select **Add rule** in the drop-down list.

- f. Select an attribute, a condition, and select or enter a value.

Example

Select **Test** as an attribute, select = as a condition operator, and enter *mytest* as a value.

Follow these steps to add a group in your query:

3. Select **Add group** in the drop-down list.



Note: A group is a collection of rules and can also contain other groups within a query. You can add a rule or a group within this group.

4. Select a condition operator.

Example

Select **Or**.

5. Select **Add rule** in the drop-down list to add a rule in the group.
6. Select the required an attribute, an operator, and enter a value for the rule.

Example

Select the **Type** attribute, = as an operator, and **Compound Test** as a value.

7. Select **Add rule** to add another rule in the group.
8. Select and enter the appropriate parameters as you did for the other rules.

Example

Select **Type** as an attribute, = as a condition, and **VU Schedule** as a value.

9. Click **Apply** if you want to apply the filter now.

You can cancel the query or save the query when the results are found.

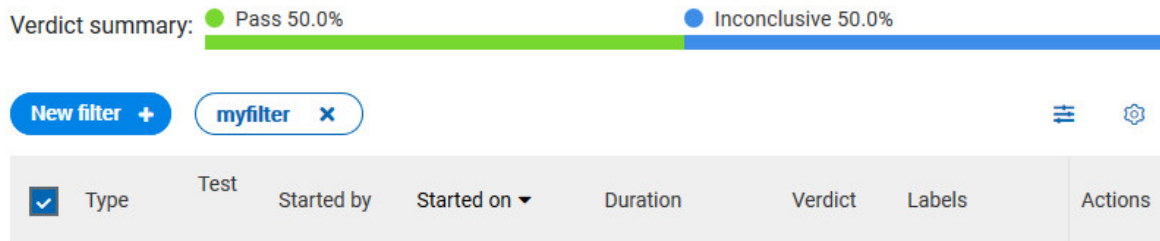
10. Click **Save**.

Proceed as follows to select your filters:

11. Enter a name for the filter and save in the dialog that is displayed.

Result

The filter is displayed as a favorite filter.



12. Click the icon to select the filters you created from the list of saved filters.

Filters

Search for filter

None
 myfilter

13. Click a filter in the list of saved filters. If a filter does not display in the list, enter the name of a filter in the search field and select it.

Note: You can delete a filter from the list or disable filters by selecting **none** in the list.

14. Apply the selected filter.

Result


The filter is displayed as a new favorite filter in the **Results** page.


Comparing test reports

After you run performance tests, you can analyze the difference between two or multiple test results in IBM® Rational® Test Automation Server. For example, you can compare the performance of an application at different time slots or different milestone builds between two test results.

About this task

You can compare the test runs for performance tests that are in the same project or in different projects.

1. Select results in the **Results** page by checking the boxes in the first column.
2. Click the  icon

 **Note:** When comparing multiple runs, you cannot compare multiple time-ranges or stages.



Result


The two or multiple test results are displayed in a browser window in the same report.

Viewing trending reports

When the performance test runs are complete, you can view the trend of response time for an application over a period of time in IBM® Rational® Test Automation Server from trending reports. In addition to the response time, you can view the trend for the loops, transactions, and performance requirements for the application.

About this task

The trending reports are available for performance tests only.

1. Select results in the first column in the **Results** page.
2. Click the  icon to view the trending reports.



Note: When comparing multiple runs, you cannot compare multiple time-ranges or stages.



Results

You can analyze the trend information for multiple test results in the same report from a browser window.

Re-executing tests from results

You can re-execute a test, a schedule, a collection of tests or schedules from IBM® Rational® Test Automation Server if you want to keep the same commit id.

Before you begin

You must have executed a test or multiple tests from the **Execution** page in IBM® Rational® Test Automation Server.

Re-executing a test from the **Results** view requires a Tester or an Owner role.

About this task

The following procedure describes how to re-execute a single test or multiple tests at a time.

Perform all the following steps from the Results page.

- Proceed as follows to re-execute one test at a time:
 1. Identify the test that you want to re-execute and click the **Re-execute** button in the Actions column.

Result

The **Execute test asset** dialog displays the same parameters as the ones that were set to execute the initial test from the **Execution** page.

2. Select another **Version** of the test.



Note: If you select another version of the test, the settings configured in the dialog and in the **Environment**, **Data Sources** and **Locations** tabs might change.

3. Modify the **Schedule** parameters.




Note: If you don't want to execute the test now, schedule the time when you want the test to be re-executed.

4. Click the **Environment** tab and select another **API test environment** if multiple environments were initially set for a test that is running an API Suite.
5. Click the **Data Sources** tab and select an override option if multiple data sources were defined for the initial test run.
6. Click the **Variables** tab and add a new variable if you want to re-execute a test asset with a different variable from the one configured in the asset.
7. Click the **Location** tab to override the default cluster location if the test asset has a docker host or static agents configured.
8. Click **Advanced** to modify the advanced parameters that were set for the initial test run.
9. Enter a new **Program Argument** if applicable to the test.



Note: If Jaeger is installed with IBM® Rational® Test Automation Server, you can enter the `-history jaeger` Program Argument to produce Jaeger traces when you run the tests.

10. Enter a new instance of **Java arguments** if applicable to the test.
11. Enter the environment variables that must be passed to the test run at runtime in the **Environment Variables** field.
12. In the **Resource Monitoring** tab, click the  , and press the Ctrl + Space keys, or enter the initial letter of a label to select a label in the list and override the current sources in the schedule.




Note: The **Resource Monitoring** tab is displayed for performance schedules and only if you have enabled the **Resource Monitoring from Service** option from Rational® Performance Tester.

The label field is grayed if you don't have any label entered in your current project. To collect data during the test execution, you must first enter labels in the Resource Monitoring sources page from IBM® Rational® Test Automation Server.



For more details about controlling Resource Monitoring sources and labels, see the links at the end of this page.

13. Click the **Re-execute**  icon to re-execute the test now or at a scheduled time.

- Proceed as follows to re-execute multiple tests at a time:

1. Click the checkboxes for the tests that you want to re-execute.

You can select them one by one or select all of them.

Result

A toolbar is displayed.

2. Click the **Re-execute** icon  in the toolbar.

Result

A message indicates the number of selected tests and the number of tests to be re-executed. The tests are re-executed with the same initial commit ids and parameters.

3. Click **Execute**.

Result

You can view the results, logs, and reports of the re-executed tests in the **Results** page.

Related information

[Adding a project on page 619](#)

[Test results and reports on page 456](#)

[Controlling resource monitoring sources in a schedule on page 521](#)

[Test run configurations on page 249](#)

Test results data in Grafana

By using IBM® Rational® Test Automation Server Results Data and Grafana, you can retrieve and explore test log data that is generated in IBM® Rational® Test Automation Server. You must install and configure the IBM® Rational® Test Automation Server Results Data plug-in in Grafana and create an instance of a data source that you use to explore your test log data.

Grafana is a visualization and analytics software. You can use Grafana to explore IBM® Rational® Test Automation Server metrics.

You must use IBM® Rational® Test Automation Server Results Data, which is a data source plug-in, to make requests to IBM® Rational® Test Automation Server and represent data in a Grafana dashboard.


Installing and configuring IBM Rational Test Automation Server Results Data

You must install and configure IBM Rational Test Automation Server Results Data that is delivered as a plug-in along with IBM® Rational® Test Automation Server to view and analyze the log data in Grafana. You can view the test log data that is generated when you run a test from IBM® Rational® Test Automation Server in a Grafana dashboard.


Before you begin

You must have configured your installation so that Grafana server can access IBM® Rational® Test Automation Server.

1. Download IBM Rational Test Automation Server Results Data plug-in from the following URL: `<your server address>/artifacts/grafana/hcltechsw-rtasresults-datasource.zip`.
2. Install IBM® Rational® Test Automation Server by extracting the compressed file into your Grafana plug-in directory.

 **Important:** Before you install IBM® Rational® Test Automation Server, you must be aware that you must use an additional helm command parameter **gateway.allowedOrigin** during the installation or you can run this command after installation too as detailed in [Additional Helm parameters on page 69](#).

3. Restart the Grafana server.
4. Enter the address of the local Grafana server in your web browser:


 **Important:** The plug-in is signed to be used with `http://localhost:3000` or `http://localhost:8080` URLs.

5. To verify that IBM Rational Test Automation Server Results Data is available in Grafana, follow these steps:
 - a. Click **Configuration > plug-in** in the left navigation panel of Grafana.
 - b. Enter **IBM Rational Test Automation Server Results Data** in the search bar in the **Plugins** tab.
 - c. Click IBM Rational Test Automation Server Results Data in the search results.

Result

The IBM Rational Test Automation Server Results Data readme page is displayed. You can find the links to Rational® Test Automation Server documentation and the description of the plug-in on this page.

6. Create an instance of data source in Grafana for IBM Rational Test Automation Server Results Data by performing the following steps.

 **Note:** Only users with the organization admin role can add data sources.

- a. Click **Configuration > Data sources** in the left navigation panel.
- b. Click **Add data source**.
- c. In the search bar, enter **IBM Rational Test Automation Server Results Data**, and then press **Enter**.

IBM Rational Test Automation Server Results Data is displayed in the search results.

d. Click **Select**.

The data source configuration panel is displayed to enter the data source settings.

e. Click **Default** if you want to define this instance of IBM Rational Test Automation Server Results Data as the default data source in Grafana.



Note: Ensure that IBM Rational Test Automation Server Results Data is accessible through a web browser.

f. Enter a name to identify the data source.

g. Enter the URL of Rational® Test Automation Server in the **Server** field.

h. Enter the token that you created from Rational® Test Automation Server to enable third-party integrations.

i. Click **Save and test**.

Result

If the test is successful, an instance of data source is created for IBM Rational Test Automation Server Results Data.

Results

IBM Rational Test Automation Server Results Data is displayed in the list of data sources in Grafana.

What to do next

You can retrieve the test log data in IBM® Rational® Test Automation Server by using queries.

Related information

[Download Grafana](#)

[Install Grafana plugins](#)

[Managing access to IBM Rational Test Automation Server on page 600](#)

[Add a data source](#)

[Explore](#)

[Viewing test results data in Grafana on page 467](#)

Viewing test results data in Grafana

After you install and configure IBM® Rational® Test Automation Server Results Data, you can retrieve and view the test log data, generated in IBM® Rational® Test Automation Server, by using queries. Then you can analyze your test log data in a graph and a table, or in a Grafana dashboard.

Before you begin

To be able to view your test log data in the IBM® Rational® Test Automation Server Results Data dashboard, you must have performed the following tasks in Rational® Test Automation Server:

- Run your tests with the appropriate option in the advanced settings to get test logs results in Rational® Test Automation Server. For more information about the procedure, refer to [Test logs on page 470](#).
- Installed the IBM® Rational® Test Automation Server Results Data plug-in.
- Run Grafana with one of these ports: `http://localhost:3000` or `http://localhost:8080`.
- Created an instance of data source for IBM® Rational® Test Automation Server Results Data. For more information about the procedure, refer to [Installing and configuring IBM Rational Test Automation Server Results Data on page 466](#).

1. Click **Configuration > Data sources** in the navigation panel in Grafana.
2. Click IBM® Rational® Test Automation Server Results Data in the list of data sources.
3. Click **Explore** in the navigation bar.
4. Enter the **Project id**.
The Project id is required.
5. Enter the **Result id**.
The Result id is optional.



Note: If you enter your project id only, you get all the results that are generated in your project.

6. Click the **Events** or the **Activities** tab according to the target you want to be returned in your query.
7. Enter the **Query Filters**.
8. Enter the **Timing value** and **Value**.
The timing values are sorted in ascending order.
The value must be numeric so that it can be used in a graph.



Tip: To find all the information needed in the **Explore** page of IBM® Rational® Test Automation Server Results Data (project and result numbers, query filters for the activities or the events, query filters, and timing values), perform one of the following procedures:

- Select a result in the **Queries** page in Rational® Test Automation Server and copy the result number, query filters for the events or the activities, query filters, and timing values. You can find the project number in the URL of the **Queries** page.
- Click **Copy curl command** in the **Queries** page in Rational® Test Automation Server and paste the curl command in a text editor. You can find all the information you need in the command.

```
curl -X 'POST' -H 'accept: application/json' -H 'Content-Type: application/json' -d '{"operator": "and", "rules": [{"field": "type", "operator": "is", "value": "JMeterAction"}]}' https://[redacted].ibm.com/rest/projects/85350/results/1171/data/views/table/elements/?fields=creationEvent.time,duration&sort=creationEvent.time.asc'
```

9. Click **Add query** if you want to combine queries from different projects, results ids, or with different properties in the query filters.
10. Click **Run query**.
11. Select the appropriate time range to explore the data in the default graph and table in Grafana.

Results

Your queries are set for your project and results, you can explore the log data.

What to do next

You can build a dashboard including graphs, charts, gauges, tables to get customized visual representations of your test result data and select options to display the test result values. See [Dashboard overview](#) for more details about dashboards in Grafana.

Viewing Surefire format reports

After you execute JMeter, JUnit, API, and Functional tests with advanced settings in IBM® Rational® Test Automation Server, you can view the test results in a Surefire format report by using dedicated URLs.

Before you begin

You must have configured your tests with one of the following parameters:

- For JMeter, JUnit, Postman, and HCL® AppScan Suites, set the **history** parameter with **cisterna** value in the **Additional Configuration Parameters** field in the **Advanced** settings .
- For API Suites, AFT Suites, Compound tests, Functional tests, and API tests, set the **-history cisterna** parameter in the **Program Arguments** field in the **Advanced** settings .

About this task

When a test run is complete, you can see the test result data in a report that is in a Surefire format. You can view this report by using dedicated URLs, and you do not have to do any import task.

1. Click **Analyze > Results** in the navigation pane.

Result

The **Results** page is displayed.

2. Click a test result and note its project identifier and test result identifier (ID) in the URL of the test log that is displayed in your browser. The URL is in the following format: `https://myProductionServer.com/#/Initial/project/<ProjectID>/analyze/test-log/<ResultID>`.

The identifiers are required to view the Surefire test logs in the following steps.

3. Enter one of the following URLs with the appropriate information in a web browser or in an external tool that you use to view Surefire reports:

◦ `/rest/projects/<ProjectID>/data/views/surefire/`

Enter your REST API address and the project identifier to view the list of all test cases in the project with details about their status.

Example: `https://myProductionServer.com/rest/projects/230200/views/surefire/`

◦ `/rest/projects/<ProjectID>/results/<ResultID>/data/views/surefire/`

Enter your REST API address, the project identifier, and the test result identifier to view the results for this test result id in the project.

Example: `https://myProductionServer.com/rest/projects/230200/results/80250/data/views/surefire/`

Results

The test result reports are displayed in a Surefire format.

Related information

[Test logs on page 470](#)

[Test log queries on page 484](#)

Test logs

When you start a run of a test asset, IBM® Rational® Test Automation Server starts to log information about the test run in a test log. You can view the test log while the test runs and after the run is completed from the **Test Log** page.

Overview

Rational® Test Automation Server captures the test log for the supported tests that you can run on Rational® Test Automation Server. For the list of supported tests, see [Suites and tests that are supported on Rational Test Automation Server on page 21](#).

Rational® Test Automation Server uses an in-built PostgreSQL database to store information about the test while the test runs on the server. Rational® Test Automation Server captures and displays the test log as the default configuration and you do not have to configure other settings during or after installation of the server.

Supported formats of test logs

The formats in which you can view the test logs are as follows:

Table 11. Supported formats for test log display


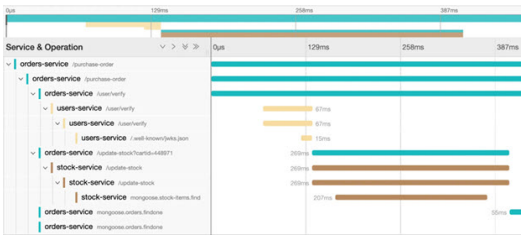
Log format	Description	Action
Default format	Displays the test log in the in-built format on the Test log page. For example, the test log can be as follows: 	You do not have to set any Program argument while you configure the test run.
Text format	Displays the test log in a plain text format and is displayed in a different browser tab or window than the one that displays the Test log page. For example, the test log can be as follows:	You must set the -history testlog Program Argument in the Advanced settings dialog while you configure the test run.

Table 11. Supported formats for test log display (continued)

Log format	Description	Action
	<pre> Name: autoResults Description: Test Version: null Verdict: fail Type: com.ibm.rational.test.lt.ltttest ----- Name: DefaultSchedule Description: Timestamp: May 25, 2022 at 9:24:10 AM GMT null Event type: com.ibm.rational.test.lt.scheduleStart Properties: Test Log Filter (ALL): 100 (ALL) Test Log Filter (WARNINGS): 100 (ALL) Test Log Filter (ERRORS): 100 (ALL) Type: start ----- Name: DefaultUserGroup Description: Timestamp: May 25, 2022 at 9:24:10 AM GMT null Event type: com.ibm.rational.test.lt.usrGroupStart Properties: Type: start ----- </pre>	
Jaeger trace	<p>Displays the test log as Jaeger traces and is displayed in a different browser tab or window than the one that displays the Test log page. For example, the test log can be as follows:</p> 	<p>Before you can configure the server to display the test log as a Jaeger trace, you must ensure that you have installed Jaeger in the cluster. See Considerations for using Jaeger traces in reports on page 205.</p> <p>You must set the -history jaeger Program Argument in the Advanced settings dialog while you configure the test run.</p>

Activities and events

The information about the tests that is collated by Rational® Test Automation Server can be classified into the following types:

Table 12. Types of information collated for test logs

Information type	Description
Activity	Specifies the events that are performed when the test runs as part of a test step. An activity comprises different events and is represented as a collection of events that were performed over time.

Table 12. Types of information collated for test logs (continued)

Information type	Description
	For example, opening a specific web page by using a web browser in the test can be an activity that comprises the different events such as connecting to the web site, sending of a request, and receiving a reply. Each of these events might take a specific amount of time, which are captured in the test log.
Event	Specifies an action performed as part of a test step. For example, start connection can be an event in a test that requires a specific web page to be opened in a web browser, as detailed in the example for an activity. The exact time at which this event occurred when the test ran is captured in the test log.

Test log tasks

You can find information about the tasks that you can perform on the the test logs in the **Test log** page. The test logs for the tests that you ran on Rational® Test Automation Server are displayed in the **Test log** page.

The tasks that you can perform on the test logs are summarized in the following table:

Table 13. Test log tasks

Task	More information
Viewing test logs after you start a test run on the Execution page.	Viewing test logs after you start a test run on page 472
Viewing test logs after the test run is completed.	Viewing test logs after the test run is completed on page 473
Viewing details about the test log.	Test log details on page 477

Viewing test logs after you start a test run

When you configure and start a test run, you can view the test log that is created progressively while the test runs.

Before you begin


You must have configured and started a test run. See [Test run configurations on page 249](#).



Note: You must have opened the **Execution** page to start the test run.

1. Click **Show in progress page** icon  in the **Actions** column of the test asset on the **Execution** page.

The asset is displayed on the **Progress** page.

2. Click the **Open action menu** icon .
3. Click the **View test log** option.

The **Test log** page is displayed with the test log collated for test run.



Note: Because the test log is collected progressively as the test runs, you must click **Refresh** to view the latest or the updated test log.

Results

You have viewed the test log for the test asset that you ran.

What to do next

You can view the details of the test log and decide on the actions you might want to take on the test log. See [Test log details on page 477](#).

Viewing test logs after the test run is completed

After you configure a test run and the run is completed, you can view the test log that is created for the test.

Before you begin

You must have configured and started a test run, and noted that the test run has the status as *Complete* displayed on the **Progress** page.

About this task


You can view the test log that is created for a specific test run from the following pages:

- The **Progress** page. To view the test log from the **Progress** page, go to [Step 1 on page 473](#).
- The **Results** page. To view the test log from the **Results** page, go to [Step 2 on page 474](#).
- The **Test log** page. To view the test log from the **Test log** page, go to [Step 3 on page 476](#).

1. Perform the following steps after the test completes its run:

- a. Go to **Execute > Progress**, if you are not already on the **Progress** page.

The test assets that are running or have completed their run are listed. If you are already on the **Progress** page, only the test asset that you selected on the **Execution** page is displayed with the status as *Complete*.

- b. Click the **Open action menu** icon  under the **Actions** column in the row of the test asset.
- c. Click the **View test log** option.

The **Test log** page is displayed with the test log collated for test run.

d. Go to [Results on page 477](#).

2. Perform the following steps after the test completes its run:

a. Go to **Analyze > Results**.

The test assets that have completed their run are listed.


b. Select the tab based on the type of test assets that you ran as indicated in the following table:










If...	Then...
You want to run AFT Suites, API Suites, Compound Tests, Rate Schedules, VU Schedules, JMeter tests, JUnit tests, or Postman tests..	Click the SUITES tab.
You ran API tests, Functional tests, or Performance tests.	Click the TESTS tab.
You want to find Suites and tests by using custom filters, when you ran Suites and tests.	Click the ADVANCED tab.


c. Identify the test asset that you ran by performing any of the following actions:




























- Scroll through the list.


 **Tip:** You can hover over the icon in the **Type** column to know the type of the test asset.

 **Note:** You can also identify the type of the asset from the icon that represents the test type as shown in the following table:


Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the AD-VANCED tab
	API test			
	Functional test			
	Performance test			



Icon	Represents	Listed in the SUITES tab	Listed in the TESTS tab	Listed in the AD-VANCED tab
	AFT Suite			
	API Suite			
	Compound Test			
	HCL AppScan CodeSweep			
	JMeter Test			
	JUnit Test			
	Postman test			
	Rate Schedule			
	VU Schedule			

- Search for the test asset by entering any text contained in the test asset name in the **Search** field.
- Click the **Filter** icon  in the **SUITES** or **TESTS** tab to filter the displayed assets based on the asset type.

For example, select **API Suite** in the **SUITES** tab to display all API Suites or select **Functional Test** in the **TESTS** tab to display all Functional tests that are in the branch.

- Click the **Filter** icon  in the **ADVANCED** tab, and then create a filter query by using the **New filter** option by performing the following steps:
 - Click **New filter**.
 - Enter a name for the filter.
 - Select an operator, and add a rule, or a group of rules.
 - Add or enter the relevant parameters and either select or enter the condition and the criteria for the condition.

You can select a parameter from the following list:

- Type
 - Test Asset Name
 - Test Asset Path
 - Last Result
 - Next Run
- v. Save the filter query to save and apply the filter query to filter the assets based on the query.

The test assets that match the filter criteria are displayed.

- Retrieve and apply a saved filter query, if you have saved filter queries previously by performing the following steps:



Note: The filter query applied previously is selected and the assets based on that filter query are displayed. To apply a different filter query, you must have created and saved the filter query.

- i. Click the **Filter** icon  in the **ADVANCED** tab.

The filter queries that you created and saved are displayed.

- ii. Click the filter that you want to apply.

The test assets that match the filter criteria are displayed.

You have identified the test asset that you ran.

- d. Click the test asset under the **Test** column.

The results and reports panel is displayed.

- e. Click the **Reports** tab, if it is not already open.

- f. Click the **Test Log** link.

The test log created for the test run is displayed in the **Test log** page.

- g. Go to [Results on page 477](#).

3. Perform the following steps after the test completes its run:

- a. Go to **Analyze > Test Log**.

The **Test log** page is displayed. The test assets that were run are displayed.

- b. Identify the test asset by its name or the type for which you want to view the test log.

- c. Click the test asset under the **Test** column.

The test log created for the test run is displayed.

d. Go to [Results on page 477](#).

Results

You have viewed the test log for the test asset that you ran.

What to do next

You can view the details of the test log and decide on the actions you might want to take on the test log. See [Test log details on page 477](#).

Test log details

You can find information about the details that are displayed for a test log on the **Test log** page. You can also find information to use the different options in the UI to change the view of the test log, find details of the events captured, navigate through the log, view or download attachments that are captured.

Before you can view the test log, you must have started a test run and opened the test log that is displayed on the **Test log** page.

An example of a test log that is displayed on the **Test log** page is as follows:

The screenshot displays the Test log interface for a test run. The top navigation bar includes 'Test' and 'Test log' links. The main header shows the test name '5cmptestsAFT' and the timestamp 'May 23rd 2022 4:38 PM'. On the right, there are 'Back to test logs' and 'Refresh' buttons. Below the header, there is a search bar and view toggles for 'Flat view' and 'Tree view'. The tree view on the left shows a hierarchy of events: 'Execution - 5cmptestsAFT' (expanded) contains 'User Group - firefox 1' (expanded), which includes 'User - [firefox 1][000001]' and five 'User Group - firefox' entries. Under 'User Group - firefox 1', there are three 'Run Status Changed' events: 'Initializing Computer(s)', 'Ramping', and 'Running'. Below these are 'UI Script' events, with 'UI Iteration - Tests/webuitest-1' expanded to show 'UI Step - Start http://www.google.com/?gws_rd=ssl' selected. The right pane shows details for the selected event: 'Time' is 'Mon, May 23rd 2022 4:42:23.379 PM'. The 'Data' section shows a 'Tech Preview' of the Google homepage. The 'Event Uid' is 'A1E938D3B637DC0F90CB73137383237'.

You can find the following details that are displayed on the **Test log** page:

Table 14. Test log details



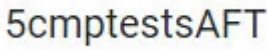
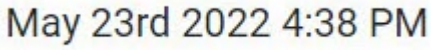
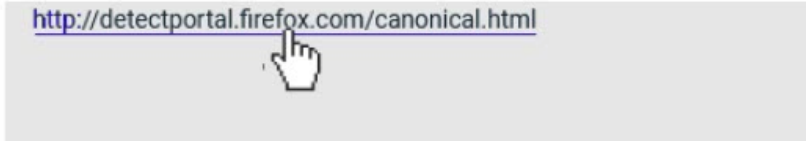
Details about...	Examples as displayed in the Test log page...
The project name	An example of the project name that is displayed is as follows: 
The name of the page	An example of the page name that is displayed is as follows:  You can click the Test log link to view the test logs of all tests that ran in your project, if you are viewing the test log of a specific test run.
The name of the test asset	An example of the name of the asset that is displayed is as follows: 
The date and time when the test run was started from the Execution page	An example of the date and time when the test run was started that is displayed is as follows: 
The activities and events captured in the test log are displayed in the Tree view in the Test log panel.	An example of the log in the tree view that is displayed is as follows:

Table 14. Test log details (continued)

Details about...	Examples as displayed in the Test log page...
<p>The link to a URL that is recorded in an event. The URL is displayed as a link. When you click this link, the specific URL opens in a web browser.</p>	<p>An example of a link recorded in the event.</p> <p>Url</p> 
<p>An attachment of an event in the test log. The details of the attachment is displayed in the Properties panel.</p>	<p>An example of a text extracted from the attachment.</p> <p>Status Line</p> <p>HTTP/1.1 200 OK</p> <p>Content Size</p> <p>89,461</p> <p>:</p> <pre>span[data-mv-comment-start]{position:relative;top:-1em}.ext-discussiontools-init-replylink-buttons{user-select:none;display:none;text-indent:0}.mv-content-ltr .ext-discussiontools-init-replylink-buttons{margin-left:0.5em}.mv-content-rtl .ext-discussiontools-init-replylink-buttons{margin-right:0.5em}.ext-discussiontools-replytool-enabled .ext-discussiontools-init-replylink-buttons{display:inline-flex}.client-nojs .ext-discussiontools-replytool-enabled .ext-discussiontools-init-replylink-buttons{display:none}.talk-overlay .ext-discussiontools-init-replylink-buttons{display:none}.ext-discussiontools-init-replylink-buttons .ext-discussiontools-init-replylink-reply{cursor:pointer}.ext-discussiontools-init-replylink-buttons .ext-discussiontools-init-replylink-bracket{color:#54595d}.mv-content-ltr .ext-discussiontools-init-replylink-</pre>
<p>The reason of a Verification Point (VP) that is captured in an event in the test log is displayed in the Properties panel.</p>	<p>An example of the reason in a VP that is displayed in the Properties panel is as follows:</p> <p>Reason</p> <p>Test failed: code expected to equal /</p> <pre>***** received : [[[200]]]</pre>

You can perform the actions detailed in the following table to change the view of the test log, find details of the events captured, navigate through the log, view or download attachments that are captured.

Table 15. Tasks that you can perform on the Test log page


Task	Action
<p>Changing the <i>Tree</i> view to a <i>Flat</i> view for the display of events in the test log panel.</p>	<p>Click  to toggle from the Tree view to the Flat view.</p>

Table 15. Tasks that you can perform on the Test log page (continued)





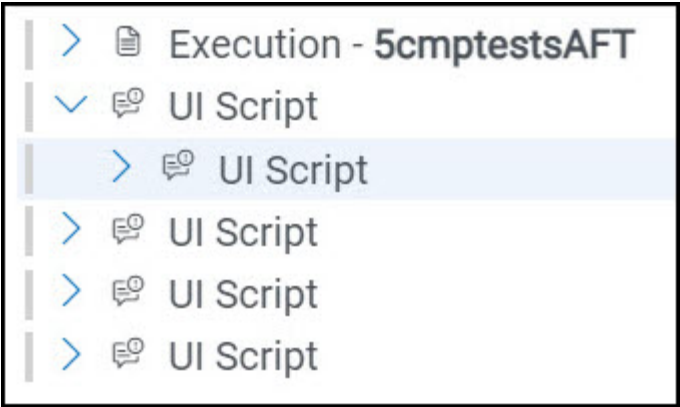
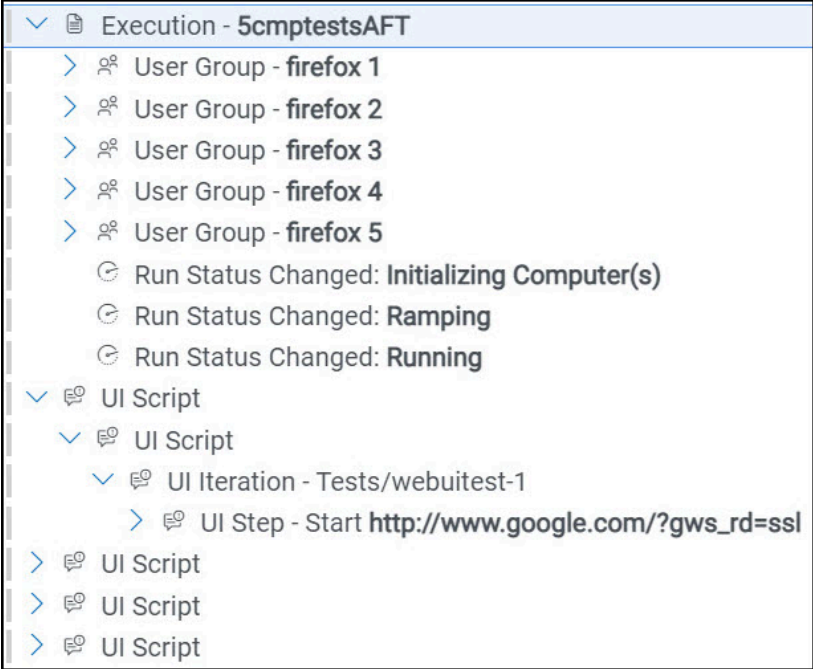
Task	Action
 Note: The <i>Tree</i> view is the default view of the events in the test log.	 Note: You can change the view from the one that is displayed by clicking  .
<p>Expanding the display of events in the test log panel.</p>	<p>Click  to expand the events that are contained within the event selected in the test log panel.</p> <p>For example, the collapsed view in the test log panel can be as follows:</p>  <p>When the event is expanded, the view in the test log panel can be as follows:</p> 

Table 15. Tasks that you can perform on the Test log page (continued)




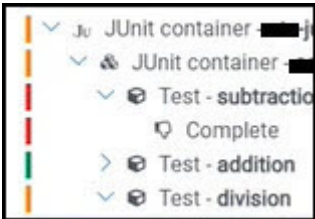


Task	Action
Collapsing the display of events in the test log panel.	<p>Click  to collapse the events that are contained within the event selected in the test log panel.</p>
Viewing primary activities when all activities are displayed.	<p>Perform the following actions in the test log panel:</p> <ol style="list-style-type: none"> 1. Click the Open action menu  icon. 2. Click Show only primary activities. <p>The primary activities captured in the test log are displayed.</p>
Viewing all activities when only primary activities are displayed.	<p>Perform the following actions in the test log panel:</p> <ol style="list-style-type: none"> 1. Click the Open action menu  icon. 2. Click Show all activities. <p>All the activities captured in the test log are displayed.</p>
Viewing failures captured in the test log.	<p>The test log displays events marked with a colored vertical bar as follows:</p> <ul style="list-style-type: none"> • A green bar is marked for an event that passed in the test step. • An orange bar is marked for an event that had an error in the test step. • A red bar is marked for an event that failed in the test step. <p>For example, the view can be as follows:</p>  <p>You can navigate up and down in the view from one error or failed event to another one by using arrows  .</p>
Searching for events in the test log.	<p>In the search bar, you can enter text to find instances of events and properties that match with the text you entered.</p>

Table 15. Tasks that you can perform on the Test log page (continued)

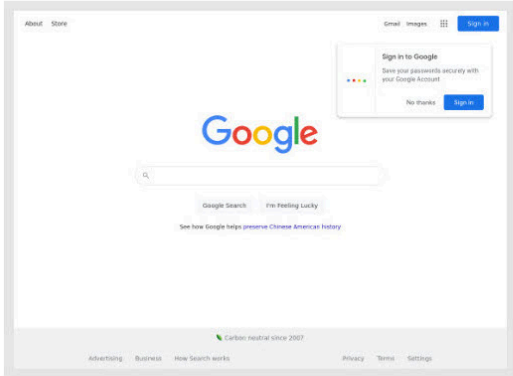

Task	Action
	<p>As a result of the search, the title of the nodes in the view and the names of the properties are highlighted. When the number of results found is displayed, you can press the F3 key to navigate from one search result to another in the tree view or flat view. You can press Ctrl and F3 to navigate from one result to another in the properties pane.</p>
<p>Viewing the properties related to the event that you select in the test log panel.</p>	<p>When you select an even in the test log panel, its properties are displayed in the properties panel.</p> <p>For example, the details that are displayed for an event can be as follows:</p> <div data-bbox="610 709 1419 1402" style="border: 1px solid black; padding: 5px;"> <p>Properties UI Step - Start http://www.google.com/?gws_rd=... Date <input checked="" type="checkbox"/> Time</p> <hr/> <p>Time</p> <p>Mon, May 23rd 2022 4:42:23.379 PM</p> <hr/> <p>Data Tech Preview</p> <div style="border: 1px solid #ccc; padding: 5px; margin: 5px 0;">  </div> <hr/> <p>Event Uid</p> <p>A1E938D3B637DCC0F90CB73137383237</p> </div> <p>You can view the following information:</p> <ul style="list-style-type: none"> • The variable in the event is displayed. • The date stamp showing the time at which the event was captured in the test log. <p>You can toggle between the calendar time and the duration from the time the test started its run in the cluster by clicking the Date and Time toggle icon . In the example, the date that is displayed in the Time field is as follows:</p>

Table 15. Tasks that you can perform on the Test log page (continued)

Task	Action
	<div data-bbox="690 310 1328 470" style="border: 1px solid black; padding: 5px; margin-bottom: 10px;"> <p>Time</p> <p>Mon, May 23rd 2022 4:42:23.379 PM</p> </div> <p>When the Time option is selected, the duration that is displayed in the Time field is as follows:</p> <div data-bbox="690 590 976 749" style="border: 1px solid black; padding: 5px; margin-bottom: 10px;"> <p>Time</p> <p>00:03:42.177</p> </div> <p>This time is the time that has elapsed from the start of the test run to the event in the test step.</p> <ul style="list-style-type: none"> • The name of the event. • The event ID. • Data that was captured as part of the test step. The items that are captured can be images, text files, PDF files, XML files, JSON files, and other supported formats. <p>You can view the attachment that is in the Data field, if it is supported to be viewed in a web browser by clicking the attachment. The attachment displays in a browser tab other than the one in which the Test log page is displayed.</p> <p>You can also copy the URL of the attachment or download a copy to your computer to use the attachment elsewhere.</p>
<p>Viewing all test logs in the Test log page.</p>	<p>When you view the test log of a specific test asset and you want to return to the Test log page to view the test logs of the other test assets, you can click the Back to test logs option.</p>

Test log queries

When a test run is complete in IBM® Rational® Test Automation Server, if a test log is generated, you can run queries to extract and view test log data. The query is used to search for the events or activities in the test logs with a project or a result scope.

The query page contains a configuration section in which you enter and select search criteria.

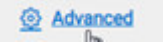
The difference between an activity and an event are the following ones:

- An activity is characterized by a duration whereas an event is characterized by a particular time.
- An activity contains events.

When you define the search criteria and run a query, the returned test log data is displayed in a table. You can manage the data in the table by adding, moving, or deleting the columns.

Prerequisites

To get result log data queries, you must configure your tests with one of the following parameters to enable the test log and data query process:

- For JUnit Suites, set the **history cisterna** parameter in the **Additional Configuration Parameters** field in the **Advanced** settings .
- For API tests and Suites, AFT Suites and Functional test, and Compound tests, set the **-history cisterna** parameter in the **Program Arguments** field in the **Advanced** settings. .



Note: When you set this parameter, you can run your queries to collect test log data into IBM® Rational® Test Automation Server.

Query scope

A **Test result query** link is available in the test reports in the **Results** page when a test log is generated. If you click the link, it opens the query page in IBM® Rational® Test Automation Server where you can view all the events or all the activities for the current test result only. In the **Scope** field of the query page, you find the result id. You can run new queries with different properties as search criteria for this result.

You can also run new queries with different properties as search criteria for all the results in a project if you open the query page from **Analyze > Queries** in the navigation pane. In this case, the **Scope** field is at the project level.

In the **Queries** page, you can perform the following actions:

- Filter the events or activities.
- Sort your query results in an ascending or a descending order.
- Add, move, or delete fields in the columns of the table that displays the data of the query.
- Save a query, rename, reset or delete a query.
- Export a query.

You can export the queries to Shell, Curl, or PowerShell scripts. You can select the script, the output format, and the number of results that are returned by the query.

Configuring and running log queries

When a test run is complete and a test log created, you can configure and run queries to view test log data in IBM® Rational® Test Automation Server for one test log result or for all the test log results in a project.

Before you begin

You must have completed the following tasks:

- Configured and run a test. See [Test log queries on page 484](#).
- Opened the Queries page from the Results page or the navigation panel.

Scope: Result 2204

▼ All events Save as

Query: Events Activities

Type:

Sort by:


Copy Curl command Add column Run Query


id	time	type	properties
102150.2204	Dec 14, 2021 11:43 AM	Namespace::createResult	
102150.2204.A	Dec 14, 2021 11:44 AM	Element::createJMeterTest	

About this task


In the **Queries** page, you can configure a query and save it. You can customize the way the log data is displayed in the results table.

1. Click **Events** or **Activites**.
2. Enter properties and operators in the **Query filters** to filter the list of events or activities.

 **Note:** You can use the **Ctrl-Space** keyboard shortcut in this field to get the list of properties and operators that you can enter as filters to create your query.

 **Example:** Select **Activities** and enter 'type is JUnitTest' in **Query filters**.

3. Click **Run Query**.
4. Perform the following steps to save a query:
 - a. Click **Save as**.
 - b. Enter a name to identify the new query and click **Save**.

 **Example:** Enter 'My query'.

Result

The new query is displayed in the events and activities list.

5. Perform any of the following actions to configure the results table:


- Enter a property followed by the **ASC** operator or **DESC** in **Sort by** to sort the data in the table.



Note: **ASC** is used to sort the data from the lowest value to the highest value and **DESC** to filter the data in a descending order. You can use the **Ctrl-Space** keyboard shortcut to assist you. If you enter a combination of sort criteria, separate them with commas.



Example: Click the **Sort by** field, press **Ctrl-Space** and select in the list 'time' and enter 'ASC'.

- Click the action menu  in a column of the table to change the column display, and then perform any of the following actions:
 - Click **Insert new column**, enter a field name in **Search new column** and click **Add**.
 - Click **Move** to move a column before or after another one in the table.
 - Click **Remove** to remove a column from the table.
- Follow these steps to add a column to the end of the table:
 - a. Click **Add column**.





Result: A window opens.

- b. Enter a field name in **Search new column**.
- c. Click **Add**.

6. Click **Save** to save the configuration of the results table.

7. Follow these procedures to manage your query after it is created:

- Click the action menu  that is next to the **Save** button and select **Reset** if you have just modified a query and you want to undo your modification.
- Click the  action menu and select **Delete** to delete the new query from the **Events and activities** list.
- Click **Copy Curl command** if you want to export a query.



Note: The curl command is copied to the clipboard. You can paste it in a command interface to use the query in an external tool.

Resource monitoring

When you apply load to a system under test, the amount of resources consumed by your system is increasing. If the capacity of the resources does not match the load, you can see performance issues in the results. Resource monitoring in Rational® Test Automation Server helps you monitoring the resources of a system and establish the

performance metrics and usage data of the system. Thus, you can observe the health of these resources while a schedule is running.

You can use resource monitoring while running a schedule to capture data, such as processor or memory usage, or to monitor the availability of hosts and services by using counters. Rational® Test Automation Server provides a comprehensive view of a system under test to help determine problems. Hosts and services can be servers, virtual machines, or any local host or network services. You can also monitor remote hosts and services with agents.

Monitoring a host or a server

You can use resource monitoring to monitor any host and network service. This method is used for host targets that the service can reach directly. With Rational® Test Automation Server, you can also monitor performance metrics that are collected and stored by a monitoring system that is monitoring a host. You can use a Prometheus server or a data collector through an exporter, OpenMetrics exporter, for example.

To monitor resources on a local system, you must add the source of the resource monitoring data from the resource monitoring page. Then you have to enter some connection settings and select the counters to monitor the sources. The metrics and usage data are exposed in a graph for each selected counter.

Monitoring a remote host with monitoring agents

You can monitor remote hosts and services with agents through wider sets of data collectors, computers, and networks. The purpose is to capture CPU load, disk space, memory, and the running process for example.

Agent-based monitoring is useful when remote services are not directly accessible from Rational® Test Automation Server through the network. Agents are closer to the target that you want to monitor. You can set up the agent on an authorized host, for example, when access to an Apache httpd or NGINX server status page or a JVM JMX port, a Prometheus server, or an OpenMetrics exporter is restricted to one or few client hosts only. The configuration task is simplified, and no security changes are required.

Agent-based monitoring is mandatory to collect data from a Windows or Linux Performance host or from a Docker host.

To monitor a host across a monitoring agent, you must first set up a resource monitoring agent on the target host for which you want to collect the performance statistics and usage data. The agent establishes a connection with Rational® Test Automation Server. When the connection is set, the agent is showing up in the list of agents on the resource monitoring main page. Then, you can select the agent in the list of target sources in the dialog **Access target from** in the dialog that opens from **Add a Source**.

Click the links in the next section of this page for more details on the resource monitoring tasks that you can perform in Rational® Test Automation Server.

Related information

[Resource monitoring capabilities on page 489](#)

[Resource monitoring agents on page 507](#)

Resource monitoring capabilities

With Rational® Test Automation Server, you can monitor a host or a service that you add to a project as a source. You can select the statistics counters that are used to collect metrics and the resource usage data throughout the monitoring system. You can visualize metrics and data that are gathered during the monitoring process in Rational® Test Automation Server.

Requirements:

- You must be a *Team Space Owner* to create, modify and remove resource monitoring sources in a team space. Any team space *Member* (not owner) can only view resource monitoring sources.
- You must be a project *Owner* to create, modify and remove resource monitoring sources in a project. Any project *Member* (not the owner) can only view resource monitoring sources.

Supported sources

With IBM® Rational® Test Automation Server, you can monitor resources for the following sources:

- **Apache httpd server:** You can optionally use agents.
- **NGINX and NGINX Plus:** You can optionally use agents.
- **Java Virtual Machine:** You can optionally use agents.

You can monitor JVM resources from a local or a remote system. Some parameters must be set in the command before running the Java Virtual Machine. For more details, see the link to the page about starting a Java Virtual Machine on this page.

- **Windows Performance host:**

To monitor the performance of a Windows host, you must have installed an agent on the target Windows host

or on a Windows host that is configured to access the performance data of the Windows host target.

- **Linux Performance host:**

To monitor the performance of a Linux host, you must have installed an agent on the target Linux host.

- **Docker host:**

A resource monitoring agent is mandatory to get a Docker data collector. The Docker host source is added to the list of resource monitoring data collectors when the agent is installed.

- **Prometheus server:**

You can monitor metrics of a host under test that are collected by a Prometheus server. Prometheus collects metrics from monitored targets by regularly requesting appropriate HTTP endpoints on these targets (called *scraping*).

The metrics data are collected through data collectors sets (groups of monitoring counters) that are tracking the system performance. The performance data results are stored in the Prometheus database so that they can be consumed by external systems through a REST API.

In Rational® Test Automation Server, you can query these performance metrics collected by Prometheus servers. 'PromQL' is the language for querying Prometheus metric data. Refer to Prometheus PromQL documentation for information about this query language.

You can select default queries, or create additional ones.

A Prometheus server is required. It must be configured to scrap a set of exporters, every 15 seconds by default. Pushgateway, service discovery, and Alertmanager are optional.

If Rational® Test Automation Server cannot directly reach the Prometheus server, you can consider setting up an agent on an authorized host or on the same host that is running Prometheus.

- **OpenMetrics exporter:**

You can monitor the metrics of a host under test that are collected and exposed through an OpenMetrics format or a Prometheus exporter format.

Rational® Test Automation Server can scrap metrics exposed by an OpenMetrics or Prometheus exporter through metric counters. No Prometheus server installation is required, you only need to set up one or multiple OpenMetrics, or Prometheus exporters that fit your software or host target.

If Rational® Test Automation Server cannot directly reach the Prometheus server, you can consider setting up an agent on an authorized host or on the same host that is running Prometheus.

Monitoring an NGINX server

To monitor an NGINX server, you must add the NGINX server source to Rational® Test Automation Server, enter connection settings and select the counters that are used to capture the monitoring usage data.

Before you begin

You must have logged in Rational® Test Automation Server and be the owner of an existing new project.

If you monitor a remote host, it must be connected with the computer that you use to access resource monitoring. See [Resource monitoring agents on page 507](#).

You can optionally use agents.

About this task

This task applies to NGINX and NGINX Plus.

If you use an agent to connect

1. Click **Infrastructure > Resource Monitoring**.

The resource monitoring page opens.

2. In the resource monitoring page, click **Add a Source** and select **Add an NGINX server...**

3. Fill in the following connection settings in the **New server** dialog:

- a. In **Target host**, enter the IP address or host name and port number of the host where the server to monitor is installed.



Note: For hosts that are already connected to Rational® Test Automation Server through an agent, you have only the **Access target from** field enabled. Move directly to step 4.

- b. In **Server edition**, select the appropriate server from the drop-down list.

- c. In **Path to the status page of the server**, enter the name of the page to view the status of the server.



Note: If you select NGINX Plus (with API version 3) as a source, you must specify the name of the path to view the API root of the NGINX Plus server.

- d. In **Security**, select the following options:

Secured with TLS/SSL

To accept the server certificate.

Trust self-signed certificate

To accept the server certificate.

Do not verify host name

To ignore verification of the host name in the certificate.

Require credentials

If the server requires log-in credentials, enter a **User name** and **Password**.

- e. Click **Add**.

The **Choose Data Counters** dialog opens.

4. Select and save the statistics counters to monitor the source. You can select them from the list where they follow the server logical organization.

For a faster selection, select the counters from the built-in sets drop-down list where they are organized by theme and save your selection.

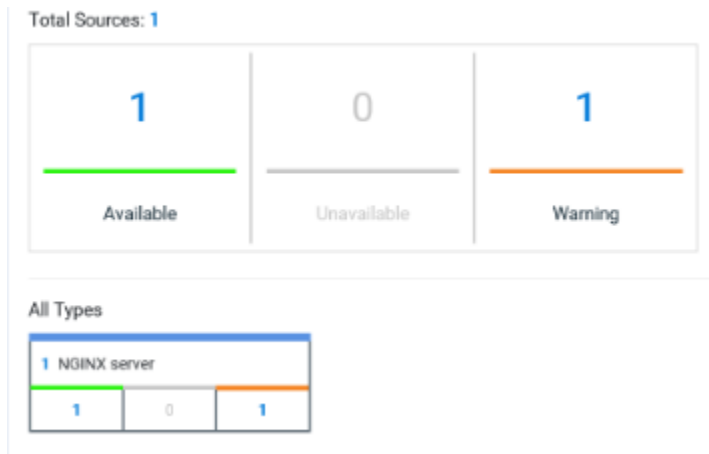
Note:

For Docker host, you can select different sets of counters.

- The first level of counters: Generic counters that are related to a Docker Image and all its running Docker Containers (in an exit, running or paused, created, restarted, removing, dead or transitive state).
- The second level of counters: Specific counters that are related to the existing Docker Containers.

Results

When the selected counters are saved, two tables are displayed in the resource monitoring main page. They contain the total number of sources you have added and the number of sources ordered by type.



What to do next

You can click the links in the tables to view the performance metrics of your monitored system.

Related information

[Viewing resource monitoring usage data on page 528](#)

Monitoring an Apache httpd server

To monitor an Apache httpd server, you must add the Apache httpd server source to Rational® Test Automation Server, enter connection settings and select the counters that are used to capture the monitoring usage data.

Before you begin

You must be logged in Rational® Test Automation Server and be the owner of an existing new project.

You can optionally use agents. See [Resource monitoring agents on page 507](#).

1. Click **Infrastructure > Resource Monitoring**.

The resource monitoring page opens. See [Resource monitoring agents on page 507](#).

2. In the resource monitoring page, click **Add a Source** and select **Add an Apache httpd server...**
3. Fill in the following connection settings in the **New server** dialog:



Note: For hosts that are already connected to Rational® Test Automation Server through an agent, you have only the **Access target from** field enabled. Move directly to step 4.

- a. In **Target host**, enter the IP address or host name and port number of the host where the server to monitor is installed.
- b. In **Server edition**, select the appropriate server from the drop-down list.
- c. In **Path to the status page of the server**, enter the name of the page to view the status of the server.
- d. In **Security**, select the following options:

Secured with TLS/SSL

If the application server is secured with TLS/SSL.

Trust self-signed certificate

To accept the server certificate.

Do not verify host name

To ignore verification of the host name in the certificate.

Require credentials

If the server requires log-in credentials, enter a **User name** and **Password**.

4. Click **Add**.

The **Choose Data Counters** dialog opens.

5. Select and save the statistics counters to monitor the source. You can select them from the list where they follow the server logical organization.

For a faster selection, select the counters from the built-in sets drop-down list where they are organized by theme and save your selection.

Results

When the selected counters are saved, two tables are displayed in the resource monitoring main page. They contain the total number of sources you have added and the number of sources ordered by type.

What to do next

You can click the links in the tables to view the performance metrics and resource monitoring usage data for the monitored server.

Related information

[Resource monitoring on page 487](#)

[Viewing resource monitoring usage data on page 528](#)

Monitoring metrics collected by a Prometheus server

You can monitor your system resources by using metrics data that are collected by a Prometheus server while executing a schedule.

Before you begin

You must have created a project in Rational® Test Automation Server.

You must have installed and configured a Prometheus server.

Read the concept information about resource monitoring and monitoring Prometheus server source. See [Resource monitoring on page 487](#)

About this task

The following procedure describes how to select and create Prometheus queries in Rational® Test Automation Server. You must set a connection between Rational® Test Automation Server and the Prometheus server first to access to the Prometheus queries.

Enter the connection settings, then select queries.

1. Click **Add a source** and select **Add a Prometheus Server**.
2. Enter the connection settings in the **New server** dialog box,.

- a. In **Target host**, enter the IP address or host name and port number of the host where the Prometheus server is installed.
- b. Enter the **Path to the API root of Prometheus** to point to the Prometheus API where the monitoring data are saved.
- c. Enter the security parameters if required. For information about this section, see related links.
- d. Click **Add** to close the dialog.

Result

The Prometheus server source is created successfully in the background and a dialog opens to select the Prometheus data queries.

3. In the **Select Prometheus queries** dialog, select queries that are available in the list to collect Prometheus metrics data by clicking the checkboxes.

You can reset the selection.

Select Prometheus queries

Choose the data to be collected from Prometheus at tp-

The screenshot shows a dialog box titled "Select Prometheus queries". At the top right, there is a blue button labeled "Add a query" with a plus icon. Below this is a table with the following columns: "Name" (with a refresh icon) and "Actions" (with eye and square icons). The table contains the following rows:

Name	Actions
<input checked="" type="checkbox"/> HTTP server requests count	
<input checked="" type="checkbox"/> HTTP server requests seconds	
<input checked="" type="checkbox"/> Node CPU busy seconds	
<input checked="" type="checkbox"/> Node disk read bytes	
<input checked="" type="checkbox"/> Node disk read time	
<input checked="" type="checkbox"/> Node disk write time	
<input checked="" type="checkbox"/> Node disk written bytes	

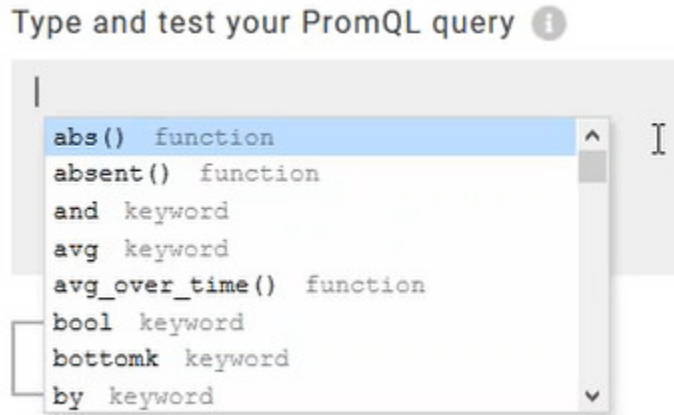
At the bottom of the dialog, there are three buttons: "Cancel", "Reset", and "Save".

4. Follow these steps to create a query:

- a. Click **Add a query**.
- b. Enter a name for the new query in the **Create a Prometheus query** dialog.
- c. Enter a PromQL query.any keyword and press Ctrl+Space in the **Type and test your PromQL query**.

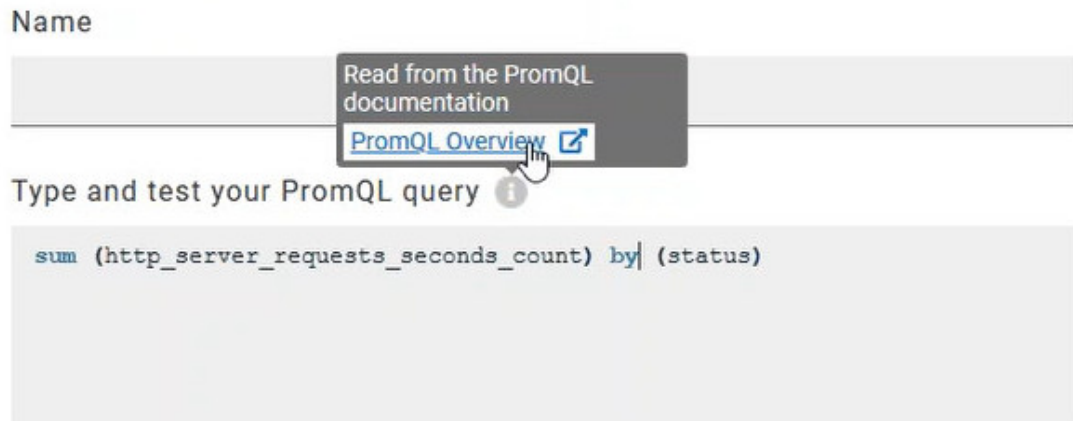
Result

All the possible combinations of keywords and metrics related to your searched keyword are listed.



d. Select a query in the list.

e. Click the link that points to the 'Querying Prometheus documentation' page in the information tool tip to find other examples of queries in a web page that you can use as samples.



f. Click **Test query**.

If the test is successful, a green check is displayed, otherwise you get a red check with an error message.

g. Click **Create**.

5. Proceed as follows to duplicate a query:

a. Click the **Duplicate** icon  to duplicate a standard query.

b. Rename the duplicated query in the **View the Prometheus query** dialog. Make sure that the name you give to the query is not already used.

c. Modify the query type.

- d. Test the query .
- e. Click **Create**.



Note: You can click Create or Duplicate only if the test of the query is successful and the name of the query is not already in use.

Create a Prometheus query


Name

JVM Memory

Type and test your PromQL query ?

```
jvm_memory_committed_bytes
```

✔

6. Proceed as follows to modify a query:
 - a. Click the **Modify** icon  .
 - b. Modify the name and type of the query.
 - c. Test the query.
 - d. Click **Update**.
 - e. Delete duplicated or created queries.
 - f. Click **Save** to close the **Select Prometheus queries** dialog.
7. Click Save and close the **Select Prometheus queries** dialog.

Results

The resource monitoring page displays the total number of sources you have added to your project and the number of sources ordered by type in cards.

What to do next

Click the links in the cards to go to the sources page and see the performance metrics for each source you added to your project. For more details, see [Viewing resource monitoring usage data on page 528](#)

Related information

[Resource monitoring on page 487](#)

[Viewing resource monitoring usage data on page 528](#)

Monitoring metrics exposed by an OpenMetrics exporter

With Rational® Test Automation Server, you can monitor your system resources by using metrics data that are exposed by an OpenMetrics or a Prometheus exporter through metric counters while executing a schedule.

Before you begin

You must have at least one exporter or software to expose monitoring metrics with an OpenMetrics or Prometheus format.

You must have created a project in Rational® Test Automation Server.

No Prometheus server installation is required, you only need to set up one or multiple OpenMetrics, or Prometheus exporters that fit your software or host target. Refer to the link at the end of this page to see the list of exporters.

About this task

The following procedure describes how to select and create OpenMetrics counters in Rational® Test Automation Server. You must set a connection between Rational® Test Automation Server and the OpenMetrics exporter to select counters.

1. Click **Add a source** and select **Add an OpenMetrics exporter** .
2. In the **New OpenMetrics exporter** dialog that opens, enter the OpenMetrics exporter details to access its exposition endpoint as follows:
 - a. In **Target host**, enter the IP address or host name and port number of the host where the server is installed.
 - b. Enter the **Path to the exposition endpoint** to point to the application where the data performance results must be stored. As the path depends on the type of exporter used, refer to the exporter's documentation and configuration to indicate the right path.
 - c. In **Security**, enter the security parameters if required.
 - d. Click **Add** to close the dialog.

Result

The source is created successfully in the background. A dialog box opens to select the OpenMetrics counters.

3. Click the checkboxes to select counters in the **Select OpenMetrics counters** dialog.
4. Follow these steps to create a counter:

- a. Click **Add a counter**.
- b. Enter a name for the new counter, in the **OpenMetrics counter**.
- c. Select metrics in the drop-down list or enter a keyword in the **Dimensions** field to enable the dynamic input help. You can also use the search field that appears in the list of metrics.

Metric

Choose a metric

Search

- http_server_requests_seconds
- http_server_requests_seconds_count
- http_server_requests_seconds_max
- http_server_requests_seconds_sum
- jvm_buffer_count_buffers

- d. Select **Dimensions** from the dynamic fields when they are required.

Metric

node_uname_info

Dimensions

domainname

Show (none)

machine

Hide x86_64

nodename

Hide matching Enter a regular expression

release

Show all 4.15.0-29-generic

sysname

Show all Linux

- e. Select the appropriate OpenMetrics counter that you are looking for.
- f. Click **Create**.

Create an OpenMetrics counter

Name

JVM Memory

Metric

jvm_memory_used_bytes

Dimensions

area


Show all heap,nonheap

id

Show all 6 items selected

Cancel Create

5. Follow these steps to duplicate a counter:

- Select a counter in the list, and click the **Duplicate** icon .
- Modify the name of the counter and the Dimensions.
- Click **Create**.

6. Proceed as follows to modify a new counter:

- Click the **Modify** icon.
- Modify the name, Metrics, and Dimensions of the counter in the **Modify the OpenMetrics counter**.
- Click **Update**.



Note: You can create, duplicate, or update a counter only if the name of the counter is not already used.

7. Click **Save** and close the **Select OpenMetrics counters** dialog.



Note: You can delete counters, but only the ones that you created or duplicated.

Results

The resource monitoring page displays the total number of sources you have added to your project and the number of sources ordered by type in cards.

What to do next

Click the links in the cards to go to the sources page and see the performance metrics for each source you added to your project. For more details, see [Viewing resource monitoring usage data on page 528](#).

Related information

[Exporters and Integrations](#)

[Resource monitoring on page 487](#)

Monitoring a Java Virtual Machine

To monitor a Java Virtual Machine, you must add the Java Virtual Machine source to the project in Rational® Test Automation Server and select the counters that are used to capture the monitoring metrics.

Before you begin

You must have completed the following tasks:

- Configured and started the Java Virtual Machine from a command to connect it Rational® Test Automation Server.
- See [Starting a Java Virtual Machine on page 518](#).
- Logged in Rational® Test Automation Server. You must be the owner of an existing new project.
- You can optionally use agents. See [Resource monitoring agents on page 507](#).

About this task

The Java Virtual Machine source is added to the list of resource monitoring sources when the agent is installed.

1. Click **Infrastructure > Resource Monitoring**.
The resource monitoring page opens.
2. In the resource monitoring page, click **Add a Source** and select **add a Java Virtual Machine**.
3. Fill in the following connection settings in the **New server** dialog:



Note: For hosts that are already connected to Rational® Test Automation Server through an agent, you have only the **Access target from** field enabled. Move directly to step 4.

- a. In **Target host**, enter the IP address or host name and port number of the host where the server to monitor is installed.
- b. In **Server edition**, select the appropriate server from the drop-down list.

c. In **Path to the status page of the server**, enter the name of the page to view the status of the server.

d. In **Security**, select the following options:

Secured with TLS/SSL

If the application server is secured with TLS/SSL.

Trust self-signed certificate

To accept the server certificate.

Do not verify host name

To ignore verification of the host name in the certificate.

Require credentials

If the server requires log-in credentials, enter a **User name** and **Password**.

4. Click **Add**.

5. Select and save the statistics counters to monitor the source. You can select them from the list where they follow the server logical organization.

For a faster selection, select the counters from the built-in sets drop-down list where they are organized by theme and save your selection.

Result

When the selected counters are saved, two tables are displayed in the resource monitoring main page. They contain the total number of sources you have added and the number of sources ordered by type.



What to do next

You can click the links in the tables to view the performance metrics and usage data in a graph for the monitored Java Virtual Machine.

Related information

[Resource monitoring on page 487](#)

[Viewing resource monitoring usage data on page 528](#)

Monitoring a Docker host

To monitor a Docker host, you must add the Docker host source to Rational® Test Automation Server, enter connection settings and select the counters that are used to capture the monitoring usage data.

Before you begin

You must have completed the following tasks:

- Logged in Rational® Test Automation Server and you must be the owner of an existing new project.
- Installed a Docker agent. A resource monitoring agent is mandatory to get a Docker data collector. [Resource monitoring agents on page 507](#).
- Started the Docker server to connect the Docker host to Rational® Test Automation Server through the agent.

About this task

The Docker host source is automatically added to the list of resource monitoring sources when the agent is installed.

1. Click **Infrastructure > Resource Monitoring**.
The resource monitoring page opens.
2. In the resource monitoring page, click **Add a Source** and select **add Docker host**.
3. Select an address to connect to the target host in **Access target from**.
4. Click **Add**.
5. Select and save the statistics counters to monitor the source. You can select them from the list where they follow the server logical organization.
For a faster selection, select the counters from the built-in sets drop-down list where they are organized by theme and save your selection. You can select different sets of counters:
 - The first level of counters contain generic counters that are related to a Docker Image and all its running Docker Containers (in an exit, running or paused, created, restarted, removing, dead or transitive state).
 - The second level of counters contains specific counters that are related to the existing Docker containers.

Result

When the selected counters are saved, two tables are displayed in the resource monitoring main page. They contain the total number of sources you have added and the number of sources ordered by type.



What to do next

You can click the links in the tables to view the performance metrics and usage data in a graph for the Docker host source.

Related information

[Resource monitoring on page 487](#)

[Viewing resource monitoring usage data on page 528](#)

Monitoring a Windows Performance host

To monitor a Windows Performance host, you must add the Windows Performance host source to the project in Rational® Test Automation Server and select the counters that are used to capture the monitoring usage data.

Before you begin

You must have completed the following tasks:

- Installed and started an agent. See [Resource monitoring agents on page 507](#).
- Logged in to Rational® Test Automation Server. You must be the owner of an existing new project.

About this task

The Windows Performance host source is added to the list of resource monitoring sources when the agent is installed.

1. Click **Infrastructure > Resource Monitoring**.
The resource monitoring page opens.
2. In the resource monitoring page, click **Add a Source** and select **add a Windows Performance host**.

3. Select an address to connect to the target host in **Access target from**.

4. Click **Add**.

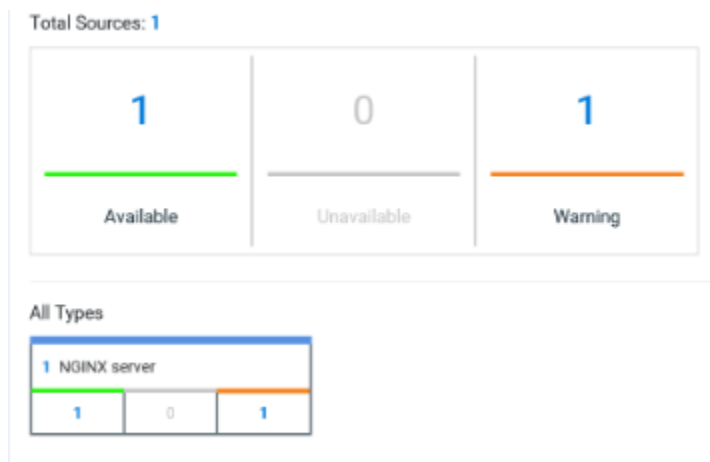
The **Choose Data Counters** dialog opens.

5. Select and save the statistics counters to monitor the source. You can select them from the list where they follow the server logical organization.

For a faster selection, select the counters from the built-in sets drop-down list where they are organized by theme and save your selection.

Result

When the selected counters are saved, two tables are displayed in the resource monitoring main page. They contain the total number of sources you have added and the number of sources ordered by type.



What to do next

You can click the links in the tables to view the performance metrics and usage data in a graph for the monitored Windows Performance host.

Related information

[Resource monitoring on page 487](#)

[Viewing resource monitoring usage data on page 528](#)

Monitoring a Linux Performance host

To monitor a Linux Performance host, you must add the Linux Performance host source to the project in Rational® Test Automation Server and select the counters that are used to capture the monitoring usage data.

Before you begin

You must have completed the following tasks:

- Installed and started an agent. See [Resource monitoring agents on page 507](#).
- Logged in to Rational® Test Automation Server. You must be the owner of an existing new project.

About this task

The Linux Performance host source is added to the list of resource monitoring sources when the agent is installed.

1. Click **Infrastructure > Resource Monitoring**.

The resource monitoring page opens.

2. In the resource monitoring page, click **Add a Source** and select **add a Linux Performance host**.

3. Select an address to connect to the target host in **Access target from**.

4. Click **Add**.

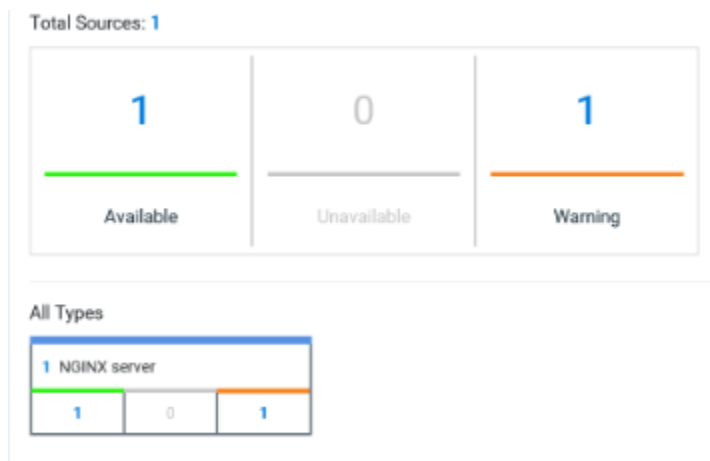
The **Choose Data Counters** dialog opens.

5. Select and save the statistics counters to monitor the source. You can select them from the list where they follow the server logical organization.

For a faster selection, select the counters from the built-in sets drop-down list where they are organized by theme and save your selection.

Result

When the selected counters are saved, two tables are displayed in the resource monitoring main page. They contain the total number of sources you have added and the number of sources ordered by type.



What to do next

You can click the links in the tables to view the performance metrics and usage data in a graph for the monitored Linux Performance host.

Related information

[Resource monitoring on page 487](#)

[Viewing resource monitoring usage data on page 528](#)

Resource monitoring agents

You can connect agents to IBM® Rational® Test Automation Server from other hosts to monitor a larger set of data collectors, computers, servers, or systems, or if a server to monitor requires authorization access. To monitor these remote servers, you must configure and run the agents so that a connection is set between IBM® Rational® Test Automation Server resource monitoring service and the agents.

Requirements:

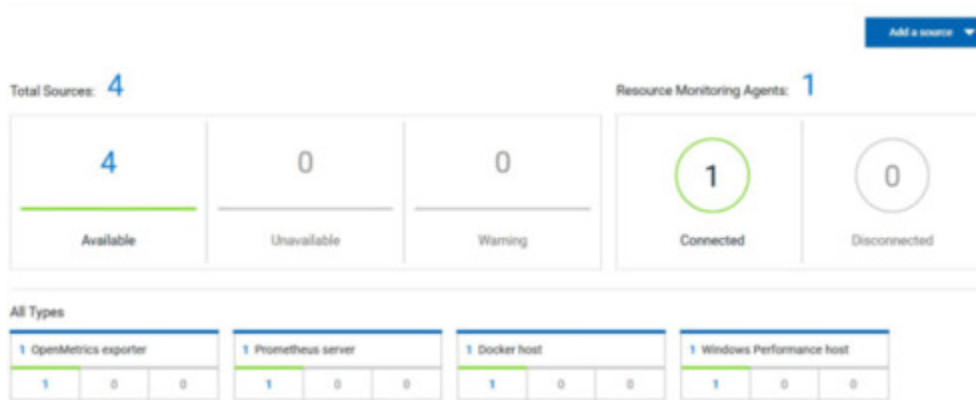
- To connect resource monitoring agents to IBM® Rational® Test Automation Server in a team space, you must be a *Team Space Owner*. Any other team space *Member* (not owner) can only view the resource monitoring agents.
- To connect resource monitoring agents to IBM® Rational® Test Automation Server in a project, you must be a project *Owner*. Any project *Member* (not the owner) can only view the resource monitoring agents.
- Agent-based monitoring is mandatory to collect data from a Windows or Linux Performance host or from a Docker host.

For testing, you might need many hosts. For example, you might have one host with the application server, another host with the database server, and some hosts to apply the user load. Due to network or firewall issues, sometimes, it becomes difficult for multiple hosts to connect to each other. Resource monitoring agents are installed on the target hosts so that they can establish a connection with IBM® Rational® Test Automation Server to gather resource statistics of the target host.

The agents always try to connect with the server through the HTTPS protocol. You must install the agent and start it. From the resource monitoring page in IBM® Rational® Test Automation Server, you can copy the command lines to download files and run the agents. When you stop the agent, the monitoring sources that you have already added persist but the live data will not be available.

You can configure your resource monitoring agent as a service so that the agent automatically runs when the host is restarted. For details, see the documentation pages about starting resource monitoring agents on Windows or Linux.

When the agent is connected to IBM® Rational® Test Automation Server, it is added to the main page of the resource monitoring service. Then, you can select the agent in the list of target sources in the dialog **Access target from** in the dialog box that opens from **Add a Source**.



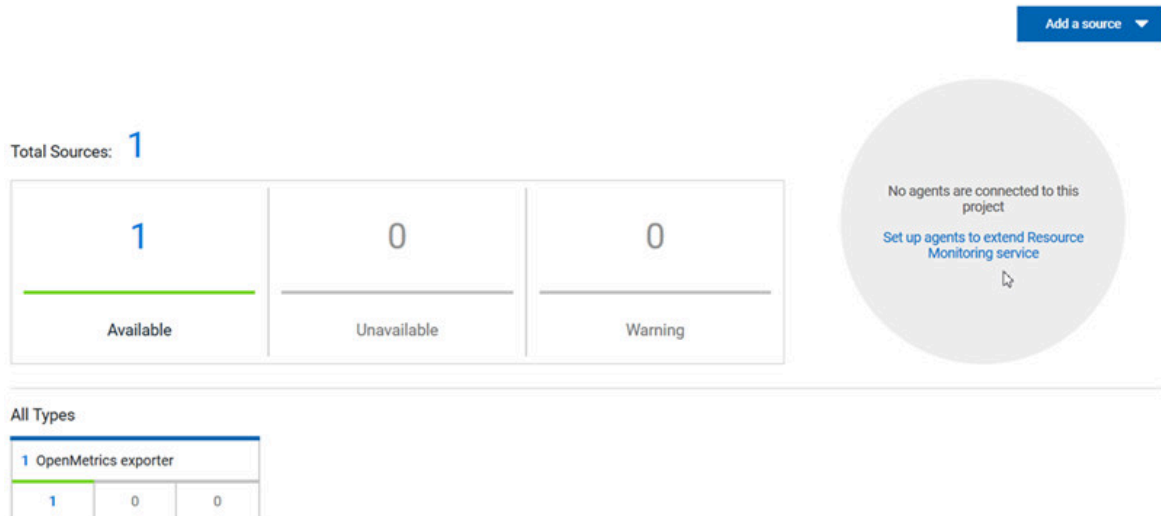
Installing and running a Docker agent on a remote host

You must install the resource monitoring Docker agents on the target host for which you want to monitor the resources and collect the metrics. You need to run an agent to establish a connection with Rational® Test Automation Server.

Before you begin

- The resource monitoring service does not require access to the agent host but the agent must have reached the service host over HTTPS.
- The Docker agent must have been launched in a Docker container and requires Docker 19.03.

1. Click **Set up Agents to extend Resource Monitoring service** link in IBM® Rational® Test Automation Server to access the agents page where you can find the instructions and commands that are to be used to install and run the agents.



Extend the Resource Monitoring service with agents ▼

- ▶ Context 👉
- ▶ Configure the Java agent
- ▼ **Configure the Docker agent**

1. Install the Docker agent
 - [Download Dockerfile](#)
 - Alternatively, you can use one of the following commands to download the Dockerfile resource:
 - ```
curl -O -J https://tp-cicd6.nonprod.hclpnp.com:/rm/agent-Dockerfile
```

📄
  - Or:
  - ```
wget --content-disposition https://tp-cicd6.nonprod.hclpnp.com:/rm/agent-Dockerfile
```

📄
2. Build the Docker image
 - Once you have downloaded the Dockerfile on your host system, launch the following command to build a dedicated Resource Monitoring Docker image:
 - ```
docker build --network=host --rm -t rmagent .
```

📄
3. Run the Docker container

2. Expand **Configure the Docker agent** and click **Download Dockerfile** to download a Docker agent. You can also use the `curl` or `wget` commands to download the agent without accessing the resource monitoring web UI.

For more facility, use the code snippets to copy and fill in the commands with the valid offline token and the Docker file name.

For Windows:

```
curl -O -J https://hostName/rm/agent-Dockerfile
```

For Linux:

```
wget --content-disposition hostName/rm/agent-Dockerfile
```

3. Copy the build command that is under **Build the Docker image**, paste it in your console and run it to build a resource monitoring Docker image.

Before you start the a Docker agent, you must build a Docker image.

```
docker build --network=host --rm -t rmagent
```

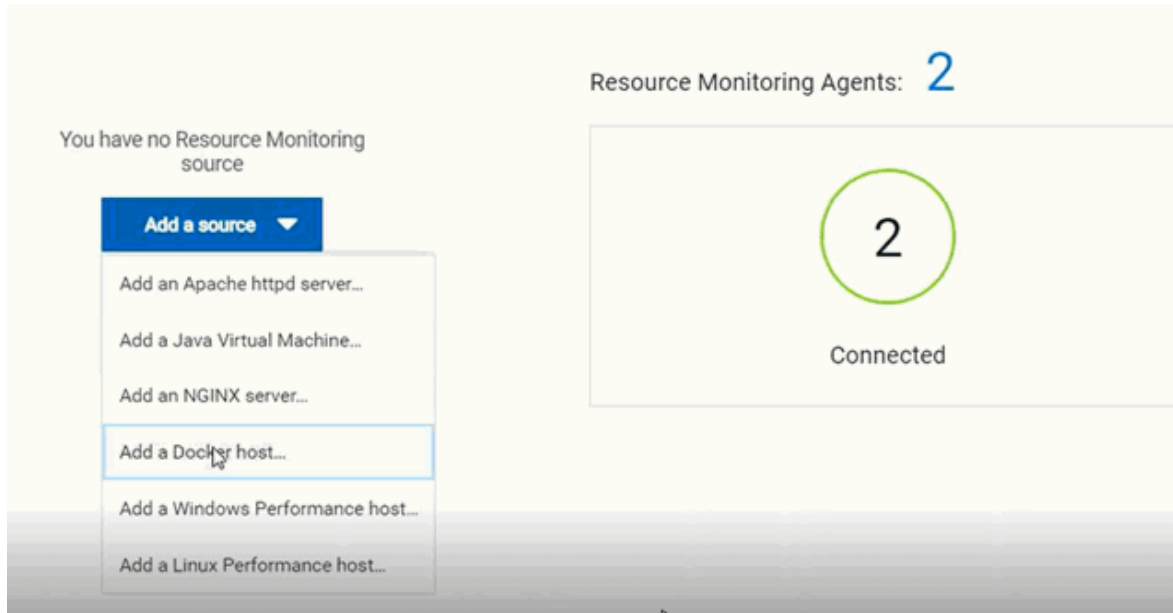
4. Follow these steps to run the agent:
  - a. Click  to fill in the appropriate command for Windows or Linux with the valid offline token.
  - b. Click  to copy the command to clipboard and paste the command in the console.
    - Command on Windows:

```
docker run -it --rm --privileged -e PROJECT_ID=230200 -e RTCP_OFFLINE_TOKEN=(Enter your offline token here) --network host -v //var/run/docker.sock:/var/run/docker.sock rmagent
```

- Command on Linux:

```
docker run -it --rm --privileged -e PROJECT_ID=230200 -e RTCP_OFFLINE_TOKEN=(Enter
your offline token here) --network host -v /var/run/docker.sock:/var/run/docker.sock
-v /etc:/etc -v /proc:/proc rmagent
```

c. Paste the command in the console and run the command.



### Results

When the agent is started, the agent is connected to IBM® Rational® Test Automation Server. The agent is displayed in the list of connected resource monitoring agents with the host name and status of the agent. The agent is also added to the main page of the resource monitoring service in IBM® Rational® Test Automation Server.

### What to do next

You can add a new resource monitoring source. It can be collected from a service or from the named agents, depending on the capabilities supported by the environment.

---

#### Related information

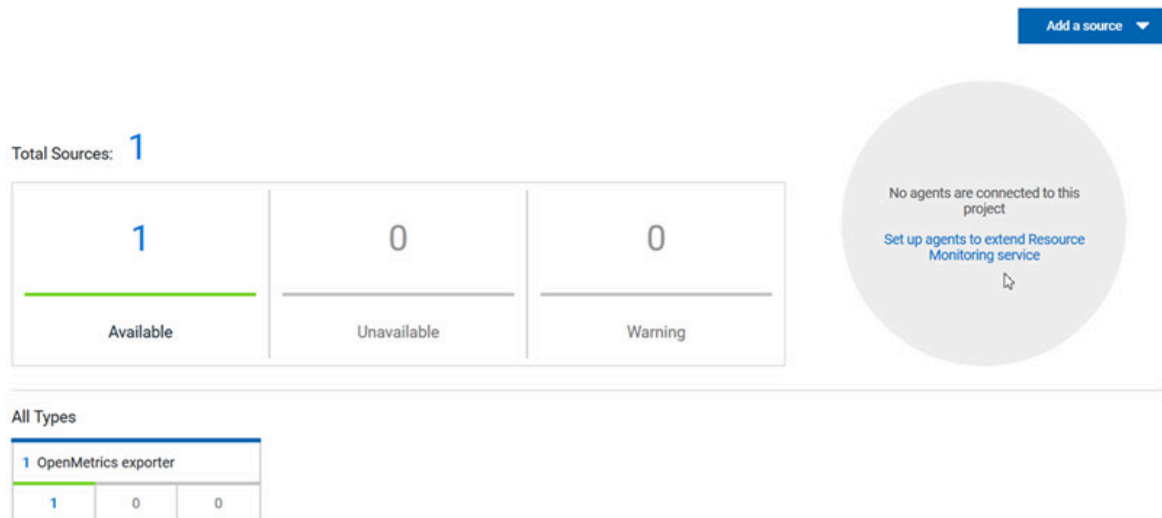
[Resource monitoring capabilities on page 489](#)

## Installing and running a Java agent on a remote host

You must install the resource monitoring java agent on the target host for which you want to monitor the resources and collect the metrics. You need to run the agent to establish a connection with Rational® Test Automation Server.

### Before you begin

- The resource monitoring service does not require access to the agent host but the agent must have reached the service host over HTTPS.
  - The Java agent must have been launched from a jar file and requires a Java 8 virtual machine.
1. Click **Set up Agents to extend Resource Monitoring service** link in IBM® Rational® Test Automation Server to access the agents page where you can find the instructions and commands that you can use to install and run the agents.



2. Expand **Configure the Java agent** and click **Download jar file** to download the Java agent. You can also use the *curl* and *wget* commands to download the agent without accessing the resource monitoring web UI. For more facility, use the code snippets to copy and fill in the commands with the valid offline token and the jar file name.

For Windows:

```
curl -O -J https://hostName/rm/Agent-jar
```

For Linux:

```
wget --content-disposition https://hostName/rm/Agent-jar
```

3. Configure the command as follows to run your agent:
  - a. Copy the appropriate command for Windows or Linux with the code snippets.
  - b. Enter the path to the directory that contains the agent .jar file, and paste the command.

#### Example

On Windows:

```
set RTCP_OFFLINE_TOKEN=(Enter your offline token here)
 java -jar (Enter the name of the downloaded jar file here)
 --ServiceUrl=https://hostName/rm --projectId=<project_id>
```

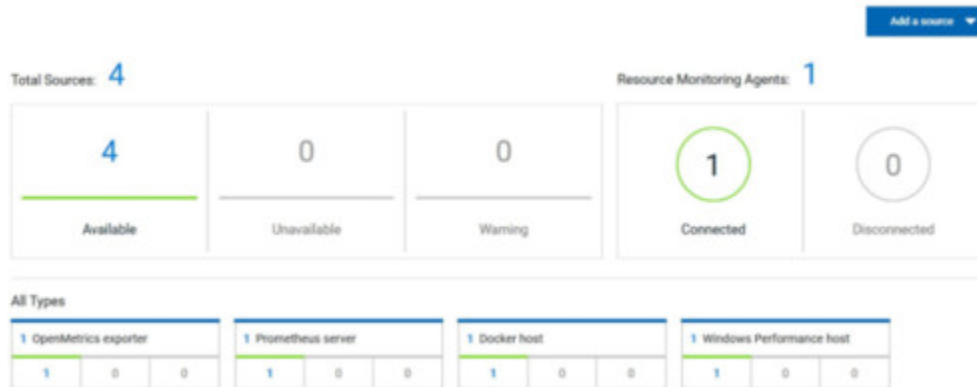
#### Example

On Linux:

```
sudo RTCP_OFFLINE_TOKEN=(Enter your offline token here) java -jar (Enter the name of the
downloaded jar file here) --ServiceUrl=portNumber/rm --projectId=<project_id>
```

## Results

When the agent is started, the agent is connected to IBM® Rational® Test Automation Server. The agent is displayed in the list of connected resource monitoring agents with the host name and status of the agent. The agent is also added to the main page of the resource monitoring service in IBM® Rational® Test Automation Server.



## What to do next

You can use another method to start an agent. See [Starting a resource monitoring Java agent as a service on Windows on page 512](#) or [Starting a resource monitoring Java agent as a service on Linux on page 515](#). You can then add a new resource monitoring source. Resource monitoring metrics can be collected from a service or from the named agents, depending on the capabilities supported by the environment.

---

### Related information

[Resource monitoring capabilities on page 489](#)

## Starting a resource monitoring Java agent as a service on Windows

To ensure that the resource monitoring Java agent starts by itself when the host is running, you can set up the environment in such a way that the resource monitoring agent can be started as a service.

### Before you begin

You must have completed the following tasks:

- Installed Java 8 on the host.
- Added IBM® Rational® Test Automation Server to the PATH environment variable.

- Created an offline token to connect the agent securely with appropriate permissions.



**Note:** You can create an offline token from the User menu of the resource monitoring page or you can re-use your active offline token. The token expires if it is not used for a month.

To allow remote access to the performance data on Windows 10, you must have considered the following information:

- The username and password are the same on the local and remote servers. Otherwise you must have provided remote server credentials.
- The remote user must be a member of the Performance Monitor Users group (start `lusrmgr.msc` and add the user to this group).
- The Remote Registry service must be running on the remote host (start `services.msc`) and must have verified the Remote Registry status).
- File and printer sharing must be enabled on the Network Interface of the remote host that is implied in the communication with the local host.
- You must have activated the following Windows Firewall rules so that the remote firewall does not block the access:
  - File and Printer Sharing (NB-Name-In)
  - File and Printer Sharing (NB-Session-In)
  - File and Printer Sharing (\*)

1. Download the latest release of `winsw`.

You can choose to download `WinSW.Net2.exe` or `WinSW.Net4.exe` depending on the version of .Net framework that you already have on the host Windows.

2. Create a folder on your local hard drive like `RMAgent-winservice`.

3. Copy the downloaded executable to this new folder and rename the file to `RMAgent-winservice.exe`.

4. Create a new text file in the same folder and name it `RMAgent-winservice.xml`.

5. Copy and adapt the following content to this new `RMAgent-winservice.xml` file to set up the offline token:

```
<service>
 <id>RMAgent-winservice</id>
 <name>Resource monitoring agent</name>
 <description>This service runs the resource monitoring agent.</description>
 <executable>java</executable>
 <env name="RTCP_OFFLINE_TOKEN" value="(Enter your offline token here)"/>
 <arguments>-jar %BASE%\RMAgent.jar --serviceUrl=https://<service-host>/rm
--projectId=<project-id>
--autoUpgradeDownloadThen=execute:cmd,/c,start,%BASE%\auto-upgrade.bat</arguments>
 <onfailure action="restart" delay="10 sec"/>
 <logmode>rotate</logmode>
</service>
```



**Notes:**



- Replace `<service-host>` by the host name of the host that runs IBM® Rational® Test Automation Server.
- Replace `<project-id>` which is the number that you find after `/projects/` in the browser's URL when you browse to this project.

6. Create a new text file in the same folder and name it `auto-upgrade.bat`.

7. Copy the following content to this new file to upgrade the agent `.jar` file automatically:

```
@echo off
for /f "tokens=*" %%a in ('dir /b /od %BASE%\com.ibm.test.rm.agent-*.jar') do set newest=%%a
%BASE%\RMAgent-winservice.exe stop
del %BASE%\RMAgent.jar
mklink %BASE%\RMAgent.jar %BASE%\%newest%
%BASE%\RMAgent-winservice.exe start
```

8. Start the command prompt as an administrator and change the directory to the newly created directory:

```
RMAgent-winservice.
```

9. Download the agent `.jar` file that is available from the agents page in the resource monitoring service, under the **Extend the Resource Monitoring service with agents** section and save it to the same directory.

10. Enter the following command in the command prompt to create a symbolic link named `RMAgent.jar` to the agent `jar` file:

```
mklink RMAgent.jar com.ibm.ibm.rm.agent-<version-and-datetime>.jar
```

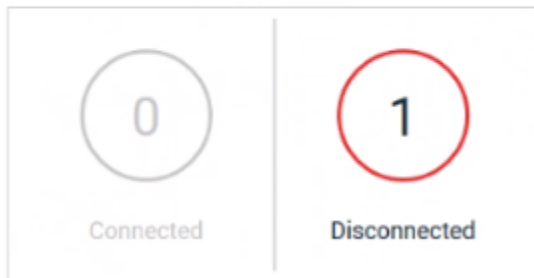
11. Enter the following commands:

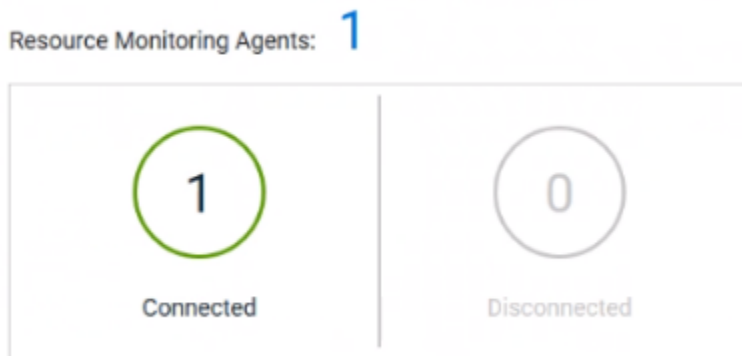
```
RMAgent-winservice install
```

```
RMAgent-winservice start
```

Add a source ▼

Resource Monitoring Agents: 1



**Notes:**

In the Services Windows application, you can check whether the service is up and running. You can see in the resource monitoring page whether the resource monitoring agent is disconnected or connected.

The resource monitoring agents write the logs to the same folder in files named `RMAgent-winservice.out.log`, `RMAgent-winservice.err.log` and `RMAgent-winservice.wrapper.log`. If the Windows service for the agent is not started or if it is not connected to the resource monitoring service, verify the log files.

**Results**

The Java agent starts as a service on Windows. When the agent is connected to IBM® Rational® Test Automation Server, it is automatically added to the list of resource monitoring sources in IBM® Rational® Test Automation Server.

**What to do next**

You can monitor the Windows agent by selecting counters. See [Monitoring a Windows Performance host on page 504](#).

**Related information**

[Other commands](#)

[winsw file](#)

[Resource monitoring capabilities on page 489](#)

**Starting a resource monitoring Java agent as a service on Linux**

To ensure that the resource monitoring Java agent starts by itself when the host is restarted, you can set up the environment in such a way that the resource monitoring agent can be started as a service.

**Before you begin**

You must have completed the following tasks:

- Installed Java 8 on the host.
- Added IBM® Rational® Test Automation Server to the PATH environment variable.
- Created an offline token to connect the agent securely with appropriate permissions.



**Note:** You can create an offline token from the User menu of the resource monitoring page or you can re-use your active offline token. The token expires if it is not used for a month.

### About this task

This topic relies on systemd services that are the default on most modern Linux distributions. Other ways may require adaptations of the instructions but the provided script will be a good basis in most cases.

1. Create a folder on your local hard drive like `/opt/RMAgent-linuxservice`.
2. Download the agent .jar file that is available from the Agents page in the resource monitoring service, below the **Extend the Resource Monitoring service with agents** section and save it to the same directory.
3. Create a new file `/etc/systemd/system/RMAgent-linuxservice.service`.
4. Add the following content to this new file:

```
[Unit]
Description = Resource monitoring agent
After = network.target

[Service]
Type = forking
ExecStart = /opt/RMAgent-linuxservice/RMAgent-linuxservice.sh start
ExecStop = /opt/RMAgent-linuxservice/RMAgent-linuxservice.sh stop
Restart = on-failure
RestartSec = 10

[Install]
WantedBy = multi-user.target
```

5. Create another file `/opt/RMAgent-linuxservice/RMAgent-linuxservice.sh`.

```
#!/bin/sh

Update the 3 following variables with the Server's host name, project id and offline token:
SERVICE_URL=https://<hostname>/rm
PROJECT_ID=<project-id>
export RTCP_OFFLINE_TOKEN=<offline-token>

ARGS="--serviceUrl=$SERVICE_URL --projectId=$PROJECT_ID --autoUpgradeDownloadThen=exitFailure"

SCRIPT=$(readlink -f "$0")
RMAAGENT_HOME=$(dirname "$SCRIPT")

Ensure we're using the latest downloaded jar file
PATH_TO_JAR=`ls -t $RMAAGENT_HOME/com.hcl.test.rm.agent-*.jar | head -1`
if [-z "$PATH_TO_JAR"]
```



```

then
 cd $RMAGENT_HOME && { curl -k -O -J $SERVICE_URL/agent-jar; cd -; }
 PATH_TO_JAR=`ls -t $RMAGENT_HOME/com.hcl.test.rm.agent-*.jar | head -1`
 if [-z "$PATH_TO_JAR"]
 then
 echo "Start the server at $SERVICE_URL to allow download of the latest agent jar file"
 echo "Exiting..."
 exit 1
 fi
fi
SERVICE_NAME="Resource Monitoring Agent"
#Pid file will reside in this script's folder
PATH_TO_PID=$RMAGENT_HOME/RMAgent-pid
#Log file will reside in this script's folder
PATH_TO_LOG=$RMAGENT_HOME/RMAgent.log

case $1 in
 start)
 echo "Starting $SERVICE_NAME ..."
 if [! -f $PATH_TO_PID]; then
 nohup java -jar $PATH_TO_JAR $ARGS >> $PATH_TO_LOG 2>&1 &
 echo $! > $PATH_TO_PID
 echo "$SERVICE_NAME started ..."
 else
 echo "$SERVICE_NAME is already running ..."
 fi
 ;;
 stop)
 if [-f $PATH_TO_PID]; then
 PID=$(cat $PATH_TO_PID);
 echo "$SERVICE_NAME stopping ..."
 kill $PID;
 echo "$SERVICE_NAME stopped ..."
 rm $PATH_TO_PID
 else
 echo "$SERVICE_NAME is not running ..."
 fi
 ;;
 restart)
 if [-f $PATH_TO_PID]; then
 PID=$(cat $PATH_TO_PID);
 echo "$SERVICE_NAME stopping ...";
 kill $PID;
 echo "$SERVICE_NAME stopped ...";
 rm $PATH_TO_PID
 echo "$SERVICE_NAME starting ..."
 nohup java -jar $PATH_TO_JAR $ARGS >> $PATH_TO_LOG 2>&1 &
 echo $! > $PATH_TO_PID
 echo "$SERVICE_NAME started ..."
 else
 echo "$SERVICE_NAME is not running ..."
 fi
 ;;
esac

```

**Notes:**



- Replace `<service-hostname>` by the host name of the host that runs IBM® Rational® Test Automation Server.
- Replace `<project-id>` which is the number you'll find after `/projects/` in the browser's URL when browsing to this project.

## Results

The resource monitoring Java agent starts automatically when the host restarts. The Linux agent is added to the main page of the resource monitoring service in IBM® Rational® Test Automation Server.

## What to do next

You can monitor the Linux Performance host source by adding counters. See [Monitoring a Linux Performance host on page 505](#).

### Related information

[systemd services](#)

[Resource monitoring capabilities on page 489](#)

## Starting a Java Virtual Machine

To monitor the sources from a Java Virtual Machine (JVM), you must first start the JVM. You must enter the IP address of the JVM, the IP port, and the security parameters in the command before running the Java Virtual Machine.

1. Enter The IP address of the JVM (local or remote host) and the IP port in the command that is used to run the Java Virtual Machine.
2. **Optional:** You can use the authentication security data to start the virtual machine. In this case, you must enter the name of the password file. You can also enter the name of an access file that might be needed if user privileges are required.

## Exemple

### Parameters used to launch a JVM without security:

```
java
-Dcom.sun.management.jmxremote
-Dcom.sun.management.jmxremote.port=9010
-Dcom.sun.management.jmxremote.local.only=false
-Dcom.sun.management.jmxremote.authenticate=false
-Dcom.sun.management.jmxremote.ssl=false
-jar MyapplicationFile.jar
```

### Parameters used to launch a JVM with authentication security:

```
java
-Dcom.sun.management.jmxremote
-Dcom.sun.management.jmxremote.port=9010
-Dcom.sun.management.jmxremote.local.only=false
```

```
-Dcom.sun.management.jmxremote.authenticate=true
-Dcom.sun.management.jmxremote.ssl=false
-Dcom.sun.management.jmxremote.password.file=jmxremote.password
-Dcom.sun.management.jmxremote.access.file=jmxremote.access (this command is optional)
-jar MyapplicationFile.jar
```

### Related information

[Resource monitoring agents on page 507](#)

[Monitoring a Java Virtual Machine on page 501](#)

## Managing sources

After you add and connect monitoring sources to your project, you might want to modify the sources in your project and the configuration of the sources.


### Before you begin


You must be a project owner or team space owner to add or delete sources.



### About this task

In the page where you can view the resource monitoring usage data and metrics, you can also manage your sources. You can modify or delete a source, add or change statistic counters, add labels to identify the sources and use them when you execute a test to collect resource monitoring results.

1. Click **Infrastructure > Resource Monitoring** in the navigation pane.
 

If you have sources and agents in your project, you can see the cards with the number of available sources and agents in your project.
2. Click a link in the card of available sources to open a list of sources.
3. Perform any of the following actions for a monitoring source:
  - Click the **Delete** icon  to delete the monitoring source and click **Delete** to confirm the deletion in the dialog box.
 

The source is removed from the list of sources.
  - Click the **Edit** icon  to change the target host settings.
 

You can change the target host address, the port, the path to the server status page and the security parameters.
  - Click the **Selection** icon  to select additional counters or change the selected counters for other ones. The changes are reflected in the resource monitoring usage data graph.
  - Click  to add a label to identify the source.
  - Click **Add a source** to add a source to your project from the list of sources.

---

### Related information

[Resource monitoring capabilities on page 489](#)

[Viewing resource monitoring usage data on page 528](#)

## Creating labels


After you connect a host or service to a project, you can add monitoring labels to identify the monitoring sources that are used when you execute a schedule from Rational® Test Automation Server. You can also use labels in any test to collect resource monitoring usage data during a test run for a defined period of time.

### Before you begin

You must have connected sources to your project in Rational® Test Automation Server. See [Resource monitoring capabilities on page 489](#).

### About this task

You must be a project owner to add labels to the resource monitoring sources in a project and you must be a team space owner to add labels in a team space.

1. Click a source that is available in your project, either in the **Total Sources** cards or in the **All Types** cards from the resource monitoring page.
2. Click  under the **Labels** column of a resource monitoring source to add a label.
3. Enter a name for the label and press the Enter key.



**Note:** You can add one label or multiple labels for the same resource monitoring source. A label can contain a maximum of 200 characters.



**Tip:** You can create the same label for different sources that belong to the same logical group to monitor the whole group with only one label.

### Results

The host source is identified by a label.

### What to do next

You can use the created labels in tests to collect resource monitoring results or persistent usage data. See [Controlling resource monitoring sources in a schedule on page 521](#) and [Collecting and viewing resource monitoring usage data on page 526](#).

## Controlling resource monitoring sources in a schedule

In Rational® Test Automation Server, you can control the resource monitoring sources that are collected in a performance schedule and override them with the labeled resource monitoring sources that are available in your current Server project.


### Before you begin

- You must have performed the following tasks in Rational® Performance Tester:
  - Created a performance schedule of **Rate** or **VU** type.
  - Added resource monitoring sources, or the labels of resource monitoring sources to a schedule. See [Adding resource monitoring sources to a performance schedule by using labels](#) .
  - Enabled Rational® Test Automation Server as resource monitoring service only if you want to replace resource monitoring labels set in Rational® Performance Tester with the labels that you created in Rational® Test Automation Server.
- You must have created a project and added resource monitoring sources to your project in Rational® Test Automation Server. See [Resource monitoring capabilities on page 489](#).

### About this task


This procedure describes how to create resource monitoring labels in Rational® Test Automation Server and how to use them to override the source or label settings related to resource monitoring in a performance schedule. The procedure applies to the following use cases:

- To replace the resource monitoring labels that were initially set in a schedule from Rational® Performance Tester with the labels that you create in Rational® Test Automation Server.
- To enable resource monitoring sources that were not enabled from Rational® Performance Tester. In that case, you can enable resource monitoring sources from the **Execution** dialog in Rational® Test Automation Server.
- To ignore resource monitoring sources that were set in the schedule and to change them for labels that match sources that are available in the Rational® Test Automation Server project.



 **Notes:** You must be a project owner to add labels to the resource monitoring sources in a project and you must be a team space owner to add labels in a team space. You must have a project owner or a tester role access to add labels to a performance schedule in a project.

1. Click a source that is available in your project, either in the **Total Sources** cards or in the **All Types** cards from the resource monitoring page.




2. Click  under the **Labels** column of a resource monitoring source to add a label.


← Sources All Types Any Status


Name	Status	Added	Observed	Labels
> OpenMetrics at to- [redacted]	available	7 hours ago	6 hours ago	SUTv2 
> Prometheus at [redacted]	available	7 hours ago	29 minutes ago	SUTv2 

3. Enter a name for the label and press the Enter key.

**Example:** Enter 'SUTv2' in three different resource monitoring sources.

 **Note:** You can add one label or multiple labels for the same resource monitoring source. A label can contain a maximum of 200 characters.

 **Tip:** You can create the same label for different sources that belong to the same logical group to monitor the whole group with only one label.

4. Click **Execution** in the navigation pane and click  on a performance schedule in the list of tests to execute a schedule.
5. Click the **Resource Monitoring** tab in the **Execute Test Asset** dialog.

 **Note:** There are different use case scenarios:



- If the **Resource Monitoring from Service** option was not enabled in Rational® Performance Tester (see Figure 1), no source is available in Rational® Test Automation Server project.

Figure 1:

**VU Schedule Details**

Category: **Resource Monitoring from Service**

Enable resource monitoring from service

A message indicates in the **Resource Monitoring** tab that Resource Monitoring was not enabled. You can run the schedule with the resource monitoring sources that are created and labeled in Rational® Test Automation Server (steps 1 to 6 in this procedure) by selecting labels that are in your Rational® Test Automation Server project (step 6).

- If the **Resource Monitoring from Service** option was enabled in Rational® Performance Tester, and data is collected from the sources you added to the performance schedule in Rational® Performance Tester (see Figure 2), the **Resource Monitoring** tab displays the list of these sources that are available in your Rational® Test Automation Server project.

Figure 2:

**VU Schedule Details**

Category: **Resource Monitoring from Service**

Enable resource monitoring from service

Collect from the following sources

Collect from sources matching at least one of the following labels

Title	Polling Time
v OpenMetrics exporter <ul style="list-style-type: none"> <li>&gt; OpenMetrics a [redacted]</li> </ul>	5 Seconds
v Prometheus server <ul style="list-style-type: none"> <li>&gt; Prometheus at [redacted]</li> </ul>	5 Seconds

To override the initial sources with the sources that you created in Rational® Test Automation Server, select the labels that are in your Rational® Test Automation Server project and that match the Server sources (step 6 in this procedure).



- If you added sources to your performance schedule by using labels in Rational® Performance Tester (see Figure 3), the **Resource Monitoring** tab displays the list of labels that are available in your Rational® Test Automation Server project.

Figure 3:

**VU Schedule Details**

Category: **Resource Monitoring from Service**

Enable resource monitoring from service

Collect from the following sources

Collect from sources matching at least one of the following labels


OpenM	Add...
Prom	
SUTv2	
	Edit...
	Remove

To select resource monitoring labels from your schedule, follow step 6 in this procedure.

In the list of sources or labels that is displayed in the **Resource Monitoring** tab:

- Green icons identify either labels that match the sources or sources that are found in the current project.
- Red icons indicate that the sources are not running,
- Yellow icons identify either labels that do not match the sources, or sources that are not found in the current project.

Data is collected for the sources that match labels only.

6. Click the , and press the Ctrl + Space keys, or enter the initial letter of a label to select a label in the list.

**Example:** Select 'SUTv3'.



ENVIRONMENT    DATA SOURCES    VARIABLES    LOCATION    **RESOURCE MONITORING**

Asset settings: list of labels

- ▲ L1\_ITS: not matching
- ▲ L2\_Proj\_ITS: not matching

To collect data during this execution, you can select existing labels that match sources:

SUTv3 x +

Label selection summary:

- ✓ SUTv3: matching
  - ✓ Prometheus at tp-junit1.nonprod.hclpnp.com:48090
  - ✓ OpenMetrics at tp-junit1.nonprod.hclpnp.com:48080 Defined in team space

Advanced    Reset    Cancel    **Execute**

You can see all the labels that you added to the resource monitoring sources in Rational® Test Automation Server.



**Note:** You can select labels matching resource monitoring sources that are created in a team space in addition to the ones that are created in a project.

7. Select the labels that match the sources.
8. Click **Execute** to execute the performance schedule.
9. Click the **Result** view in the navigation bar of Rational® Test Automation Server when the schedule is complete.
10. Expand the performance schedule result to see the resource monitoring label name in the **Details** card.

The **Details** card displays the resource monitoring labels that are used in the schedule to override the initial labels and collect resource monitoring sources (in the example, the 'SUTv3' label). For other details about the **Results** page, see the link at the end of this page.

SchedListOfRMSourcesEnabled    dilip    Sep 8, 2020    00:04:25.153    Pass

proj\_RM\_1011/Schedules/SchedListOfRMSourcesEnabled.tes...    10:16 AM

**DETAILS**

Id: 1751  
 Branch: Dilip\_1011  
 Result: SchedListOfRMSourcesEnabled  
 Finish on: Sep 8, 2020 10:20 AM  
 Status: Complete  
 Git repo: https://...testingproducts/PT\_SVT\_TestAsset.git  
 Git path: proj\_RM\_1011/Schedules/SchedListOfRMSourcesEnabled.testsuite  
 Git commit: commit sch with RM sources CRC server (committed 22 days ago) (#refs/heads/Dilip\_1011) d24e819 <>

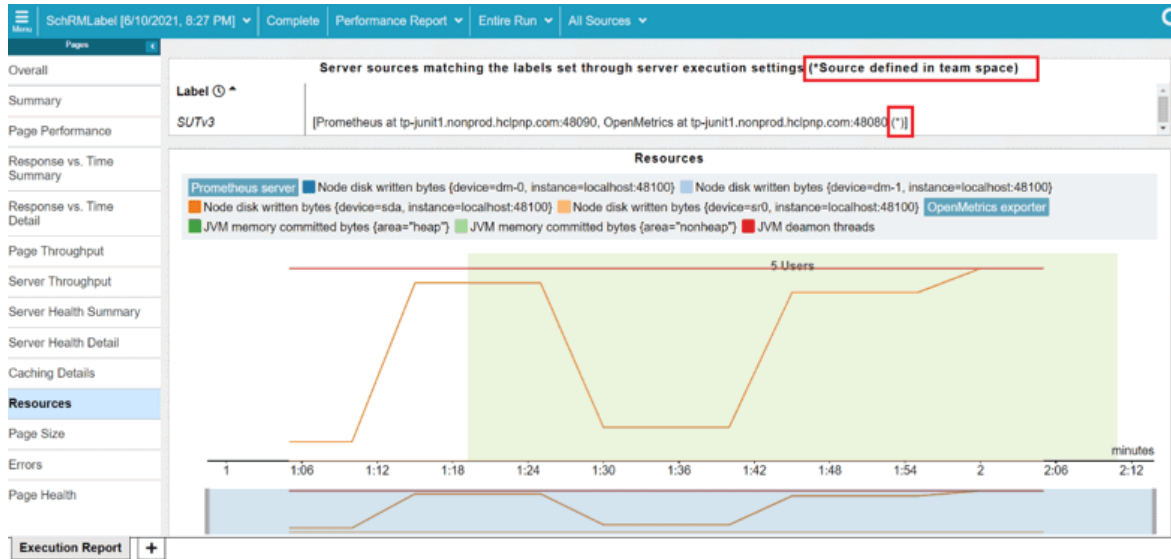
Execution location: Default Cluster  
 Resource Monitoring labels (override): SUTv3

**Reports**

- Statistics Report
- Test Log (Text)

In the **Report** card, you find the link to the Statistics Report that is generated. In the **Resource** tab of the Statistics Report, you can view the information related to the Server sources matching the labels set in the schedule.

11. Click the **Statistics Report** link in the **Reports** card to see the performance metrics of the monitoring sources matching the selected labels.



Example: You can see the statistics for the sources tagged with the 'SUTv3' label.



**Note:** In the Resources report, each resource name followed by an asterisk identifies a resource monitoring source that is defined in a team space.

#### Related information

[Test results and reports on page 456](#)

## Collecting and viewing resource monitoring usage data

With Rational® Test Automation Server, you can collect resource monitoring data in a timeframe and view the resource monitoring usage data in a graph while the test is running. You can also create queries to retrieve the resource monitoring usage data.

### About this task

While monitoring a system, you can view the current data in a graph but you might also want to view persistent data that is collected during a test run in a timeframe. To view the persistent data, you can configure the test with additional parameters before the test execution. While the test is running, you can view the resource monitoring usage data in a graph. When the test is running or complete, you can create queries to retrieve the collected data results.

To collect and view resource monitoring data, perform these tasks:

1. Configure a test with two additional parameters in the advanced settings.
2. View the resource monitoring usage data in a graph.
3. Create queries to retrieve the resource monitoring usage data.

---

Related information

[Configuring a test on page 527](#)

[Viewing resource monitoring usage data on page 528](#)

[Creating queries for resource monitoring usage data on page 530](#)

## Configuring a test

You must configure a test with two parameters to collect resource monitoring usage data in a timeframe when the test is running.


### Before you begin

You must have completed the following tasks:



- Added and configured a source.
- Created resource monitoring labels for your source.

### About this task

You can collect resource monitoring usage data during a test run for all types of monitoring sources, and all test types, except API stubs.

1. Click **Execution**, in the navigation pane.
2. Select the test that you want to run and click  in the **Actions** column to open the **Execute test asset** window.
3. Click **Advanced settings**.
4. Enter the following parameters and values in **Additional Configuration Parameters**:

Parameter name	Value
<b>rm.log.labels</b>	Enter a label name in the second edit field as the parameter value.  If you enter multiple labels, enter a comma to separate them in the list.

Parameter name	Value
	<p> <b>Example:</b> For two labels with the following names: 'LabelA' and 'LabelB':</p> <pre data-bbox="930 390 1224 415">rm.log.labels LabelA,LabelB</pre> <p>If a label contains a comma in its name, you must add two commas in order to collect the resource monitoring usage data.</p> <p> <b>Example:</b> For a label named 'Label1A,Label1B', enter the following parameter and value:</p> <pre data-bbox="930 789 1256 814">rm.log.labels Label1A,,Label1B</pre>
<p><b>rm.log.period.seconds</b></p>	<p>Enter a sampling period in the second edit field as the parameter value. The minimum period is one second.</p>

5. Click **Add** for each parameter.

6. Click **Execute**.

### Results

The monitoring data is collected while your test is running. You can see the collected data in a graph of the resource monitoring page, even 3 minutes after the test is complete.

### What to do next

You can view the resource usage data on the resource monitoring page. See [Viewing resource monitoring usage data on page 528](#).

## Viewing resource monitoring usage data

After you add and configure monitoring sources in Rational® Test Automation Server, you can view the resource monitoring usage data in a graph for each monitored source and for each selected counter.

### Before you begin

You must have completed the following tasks:

- Added and connected a monitoring source, and an agent when required, to your project to be able to view the list of available sources and agents, and the resource monitoring usage data in your project.
- Configured a test with advanced parameters to view persistent usage data in a graph.

### About this task

After you add a monitoring source to Rational® Test Automation Server, configure the source and select monitoring counters, you can view the current resource monitoring usage data for each selected counter in a graph. If you configure the test to collect data, you can also view the resource monitoring usage data that is collected for a defined timeframe. In this case, the data is still visible in the graph three minutes after the test run is complete. After this time, you can create queries to retrieve the data.

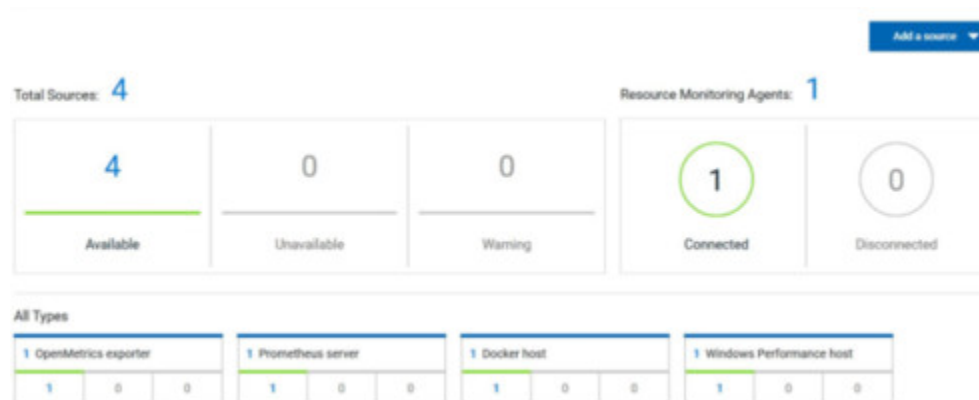


**Note:** For a Prometheus source, as data comes from the Prometheus database, it no longer shows up in the graph when the test run is complete. However, the data is collected and shows up in the queries if it matches a label.

1. Click a link either in the **Total Sources** cards or in the **All Types** cards from the resource monitoring page to access the resource monitoring usage data.

The tables display the total number of sources in your project and the number of sources by type. You can also view the number of available and unavailable sources, and the number of sources that contain warnings. If you have added '4' sources to your project, for example, click '4' in the table to access the page where you can see the resource usage data for each source.


If you have added agents to your project, you can also view the number of agents that are connected and disconnected to your project.



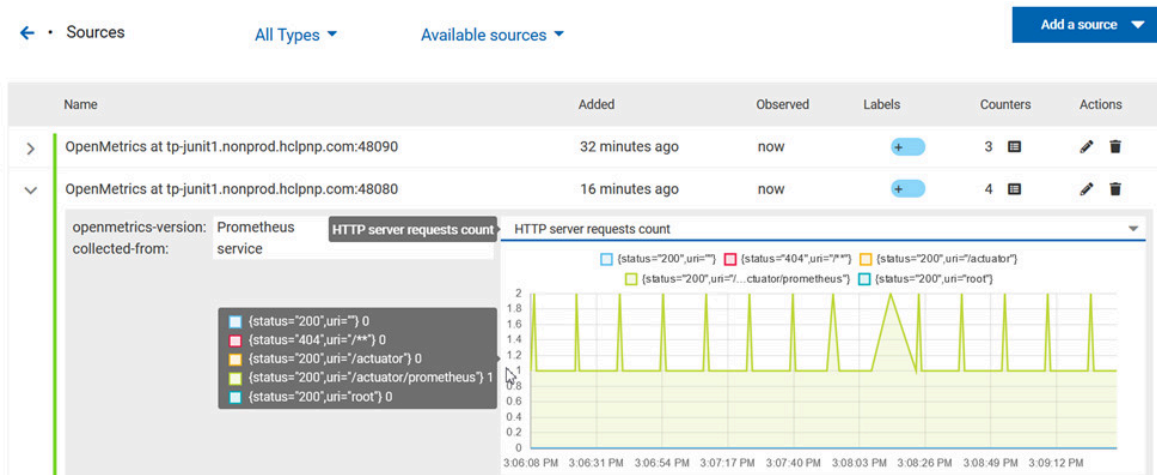
## Result

The list of sources is displayed in a new page.

You can filter the list of sources by collector type or by status.

2. Expand a monitoring source under the **Name** column to see the current monitoring data usage in a graph.
3. Click the down arrow  to select a counter in the list.

A graph is displayed for each counter.



### What to do next

From the page where you can view the resource monitoring usage data, you can also manage your sources. See [Managing sources on page 519](#).

#### Related information

[Collecting and viewing resource monitoring usage data on page 526](#)

[Creating queries for resource monitoring usage data on page 530](#)

## Creating queries for resource monitoring usage data

After you execute a test to collect resource monitoring usage data, you can create queries to retrieve the data.

### Before you begin

You must have added resource monitoring sources to a project.

### About this task

You can no longer view the monitoring usage data in graphs 3 minutes after completion of the test. However, you can create queries to retrieve the list of activities and events in your tests. You can use the following procedure as an example to create your own queries.

1. Follow these steps to filter the list of activities that are collected for a duration of less than 300 seconds:
  - a. Click **Activities**.
  - b. Enter 'Type is RMsources and duration < 300' in the **Query filters** field.
  - c. Enter 'time ASC' in the **Sort by** field to sort data by ascending order.
  - d. Click **Run Query**.
  - e. Click **Save as**.
  - f. Enter a name to identify the new query and click **Save**.

The list of activities that are collected for a duration of less than 300 seconds is displayed in the query result table, and this list is sorted in an ascending order.

2. Follow these steps to filter the list of observations that are performed during a test execution and defined by the test result identifier:

- a. Click **Events**.
- b. Enter the following filters in the **Query filters** field:

```
type is observation and anyParentActivity matches (anyChildActivity matches (requesterNature = 'test-result' and requesterId=1202))
```

- c. Enter 'time ASC, parentActivity.description ASC, specifier ASC' in the **Sort by** field to sort data by ascending order.





**Note:** You can use the **Ctrl-Space** keyboard shortcut to enter the properties and operators.


- d. Click **Run Query**.
- e. Click **Save as**.
- f. Enter a name to identify the new query, and then click **Save**.

The new query results are displayed in the table of activities and events.

3. Follow these steps to manage your query:

- Click the action menu  that is next to the **Save** button and select **Reset** if you have just modified a query and you want to undo your modification.
- Click the  action menu and select **Delete** to delete the new query from the **Events and activities** queries list.

4. Follow any of these steps to configure the query result table:

- Click the action menu  in a column of the table to change the column display, and then perform any of the following actions:
  - Click **Insert new column**, enter a field name in **Search new column** and click **Add**.
  - Click **Move** to move a column in the table.
  - Click **Remove** to remove a column from the table.
- Follow these steps to add a column at the end of the table:
  - a. Click **Add column**.



**Result:** A window opens.

- b. Enter a field name in **Search new column**.
- c. Click **Add**.

5. Click **Save** to save the configuration of the results table.

## Results

You have created two queries in your project to display:

- The list of activities that are collected for a duration of less than 300 seconds.
- The list of observations that are performed during a test execution and defined by the test result identifier.

---

Related information

[Configuring and running log queries on page 485](#)

## Configuration of a change management system

When you use any application to create and track defects (bugs), issues, or other work items, you can configure the application as a change management system on Rational® Test Automation Server. You can then create defects (bugs), issues, or other work items without the need to open your application.

You use Rational® Test Automation Server to run tests for the software that you develop and view their results. When you discover that you might want to raise defects (bugs), issues, or other types of work items for these tests, you are required to open the application that you use. You can only then, create defects (bugs), issues, or other work items in your application.

When you configure your application as a change management system on Rational® Test Automation Server, you can create defects (bugs), issues, or other work items without the need to open your application.

You can find details in the following table about the usage of the applications that you can configure as a change management system on Rational® Test Automation Server.



**Note:** You can configure only one application as the change management system on Rational® Test Automation Server.

Issue tracking application	Usage as a change management system	Go to...
Atlassian Jira Software	You can create stories, defects, or tasks as issues in your project in Jira from Rational® Test Automation Server and also launch your project in Jira from Rational® Test Automation Server to track and manage these issues.	<a href="#">Configuration of Atlassian Jira as a change management system on page 533</a>
Azure DevOps	You can create work items in your project in Azure DevOps from Rational® Test Automation Server and also launch your project in Azure DevOps from Rational® Test Automation Server to track and manage these work items.	Configuration of Azure DevOps as a change management system



Issue tracking application	Usage as a change management system	Go to...
HCL® Compass	You can use HCL® Compass as a change management system in Rational® Test Automation Server. You can also launch HCL® Compass from Rational® Test Automation Server to track and manage change requests and other records.	Configuration of HCL Compass as a change management system

## Configuration of Atlassian Jira as a change management system

You can use Atlassian Jira as a change management system in Rational® Test Automation Server to create issues in the Jira project for the tests that ran in Rational® Test Automation Server. Links to the Jira issues are displayed on the **Results** page in Rational® Test Automation Server, and you can then view and track the Jira issues in Jira.

### Prerequisites

Before you configure Jira as a change management system on Rational® Test Automation Server, you must have completed the following tasks:

- Installed Jira Software. See [Supported installations on page 533](#).
- Ensured that you obtained the access and authentication details to Jira from the Jira administrator.
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created a project in your team space in Rational® Test Automation Server. See [Adding a project on page 619](#).

### Supported installations

The following types of Jira Software installations are supported that can be configured as a change management system on Rational® Test Automation Server:

- On-premise Jira Software installations.
- On-cloud Jira Software installations.

### Task flow

You can follow the tasks that are listed in the following table when you want to use Jira as a change management system on Rational® Test Automation Server. The table also provides you the links to the information about the tasks.

Task	Go to...
Add a Jira Software server URL on the <b>Integration</b> page of Rational® Test Automation Server, and then generate	<a href="#">Adding a Jira Software server URL on page 534</a>

Task	Go to...
the API keys that authenticate Rational® Test Automation Server as an application in Jira.	
Set up an application link for Rational® Test Automation Server in the Jira server.	<a href="#">Setting up an application link in the Jira server on page 535</a>
Configure the Jira server as a change management system on Rational® Test Automation Server.	<a href="#">Configuring the Jira server as a change management system on page 537</a>
Create defects or issues for tests that ran on Rational® Test Automation Server.	<a href="#">Creating Jira issues for tests that ran on page 538</a>

You can manage the Jira server settings that you configured in Rational® Test Automation Server. You can edit the saved configuration or you can delete the configuration. For more information, see [Managing the Jira server configuration on page 542](#).

## Adding a Jira Software server URL

You must first add the URL of the server on which you have installed Jira Software on the **Integration** page of Rational® Test Automation Server. You must copy the API keys that are generated by Rational® Test Automation Server to be used to authenticate Rational® Test Automation Server as the application to connect in Jira.

### Before you begin

You must have completed the following tasks:

- Read about the prerequisite tasks in [Prerequisites on page 533](#).
- Verified that you can access the Jira server.
- Ensured that you are assigned a role of a *Project Owner* in the project. See [Managing access to server projects on page 627](#).
- Created a project in your team space in Rational® Test Automation Server. See [Adding a project on page 619](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Click **Active projects > My projects > *project\_name*** to open the project that contains the test assets.

The **Overview** page of the project is displayed.

3. Click **Manage > Integration** in the navigation pane.

The **Integration** page is displayed.



4. Click the **CHANGE MANAGEMENT SYSTEM** tab.

5. Click **Add New**, and then select the **Jira Software** option from the list.

The **Add Jira server** dialog is displayed.

6. Enter the URL of the Jira server in the **URL of the Jira server** field, and then click **Add**.

Rational® Test Automation Server verifies the connection with the Jira server and generates the API keys, which are displayed on the **Copy the generated keys** dialog.

7. Click the **Copy** icon  in the **Public Key** field to copy the key, and then paste the copied key into a text file.
8. Click the **Copy** icon  in the **Consumer Key** field to copy the key, and then paste the copied key into a text file.

## Results

You added the URL of the Jira server and generated the public and consumer keys that you must use to configure Rational® Test Automation Server as an application in the Jira server. You can view that the URL of the Jira server is displayed on the **CHANGE MANAGEMENT SYSTEM** tab and the status is displayed as *Not configured*.

## What to do next

You must share the public and consumer keys with the administrator of the Jira server to set up an application link in Jira to Rational® Test Automation Server. However, if you are an administrator of the Jira server, you can set up an application link in the Jira server. See [Setting up an application link in the Jira server on page 535](#).

## Setting up an application link in the Jira server

You must set up an application link in the Jira server to establish a connection between the Jira server and Rational® Test Automation Server.

### Before you begin

You must have completed the following tasks:

- Ensured that you are assigned the privileges of an administrator in Jira.
- Generated a public key and a consumer key for the project in Rational® Test Automation Server and saved the keys in a text file. See [Adding a Jira Software server URL on page 534](#).
- Copied the URL of Rational® Test Automation Server.

1. Log in to the Jira server.
2. Select any of the following options based on the method that you used to install the Jira server:
  - If you installed the Jira server on cloud, go to [Step 3 on page 535](#).
  - If you installed the on-premise Jira server 8.21.x or earlier, go to [Step 4 on page 536](#).
  - If you installed the on-premise Jira server 8.22.x or later, go to [Step 5 on page 536](#).
3. Perform the following steps to create the application link in the Jira server that is installed on cloud:

- a. Click **Settings > Products**.
  - b. Click **Application links** in the left navigation.
  - c. Click **Create link**.
  - d. Select **Direct application link**.
  - e. Enter the URL of Rational® Test Automation Server in the **Application URL** field, and then click **Continue**.
  - f. Verify the URL on the **Configure Application URL** dialog, and then click **Continue**.
  - g. Go to Step 6 on page 536.
4. Perform the following steps to create the application link in the on-premise Jira server 8.21.x or earlier:
    - a. Click **Settings > Applications**
    - b. Click **Application links** in the left navigation.
    - c. Verify the URL on the **Configure Application URL** dialog, and then click **Continue**.
    - d. Go to Step 6 on page 536.
  5. Perform the following steps to create the application link in the on-premise Jira server 8.22.x or later:
    - a. Click **Settings > Applications**
    - b. Click **Application links** in the left navigation.
    - c. Click **Create link**, and then select **Atlassian product** as application type.
    - d. Verify the URL on the **Configure Application URL** dialog, and then click **Continue**.
    - e. Go to Step 6 on page 536.
  6. Perform the following steps on the **Review link** dialog to provide the information about Rational® Test Automation Server:
    - a. Enter a name for Rational® Test Automation Server in the **Application Name** field.  
 You must provide a unique name to Rational® Test Automation Server. For example, `rtas`.
    - b. Select **Generic Application** as **Application Type** from the list.
    - c. Select the **Create incoming link** checkbox to set up a one-way incoming link from Rational® Test Automation Server to the Jira server.
    - d. Click **Continue**.
    - e. Enter the consumer key that was generated on Rational® Test Automation Server and copied into the text file, in the **Consumer Key** field.
    - f. Enter a name in the **Consumer Name** field.  
 For example, `IBM`.
    - g. Enter the public key from the text file in the **Public Key** field.
  7. Click **Continue**.

The link to the application you configured in Jira is displayed on the **Application links** page.

## Results

You have set up the application link to Rational® Test Automation Server in the Jira server.

### What to do next

You can configure the Jira server in your project on Rational® Test Automation Server. See [Configuring the Jira server as a change management system on page 537](#).

## Configuring the Jira server as a change management system

You must configure the Jira server as a change management system in your project in Rational® Test Automation Server before you can create issues in the Jira project for the tests that ran.

### Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Generated a public key and a consumer key for the project in Rational® Test Automation Server and saved the keys in a text file. See [Adding a Jira Software server URL on page 534](#).
- Shared the public key and consumer key with the administrator of the Jira server to set up an application link in Jira, if you are not an administrator of the Jira server.
- Created an application link in Jira, if you possess the privileges as an administrator of the Jira server. See [Setting up an application link in the Jira server on page 535](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Click **Active projects > My projects > *project\_name*** to open the project that contains the test assets.

The **Overview** page of the project is displayed.

3. Click **Manage > Integration** in the navigation pane.

The **Integration** page is displayed.

4. Click the **CHANGE MANAGEMENT SYSTEM** tab.

5. Click the **Open action menu** icon , and then click **Configure**.

The **Configure Jira server to authorize the application** dialog is displayed.

6. Perform the following steps:

- a. Click the URL of the Jira server in the dialog.

The **Welcome to Jira** page is displayed in a browser.

- b. Read the message that is displayed, and then click **Allow** to authorize the Jira server account.

After the authorization is successful, the **Access Approved** page is displayed with a verification code.

c. Copy the verification code, and then enter the code in the field on the **Configure Jira server to authorize the application** dialog that is displayed in the **CHANGE MANAGEMENT SYSTEM** tab.

d. Click **Save**.

The **Select project in Jira** dialog is displayed.

e. Select the project in Jira from the list under which you want to create defects.

7. Click **Update**.

## Results

You configured Jira as a change management system in Rational® Test Automation Server.

## What to do next

You can perform the following tasks:

- View the following information on the **CHANGE MANAGEMENT SYSTEM** tab about the Jira configuration:
  - The URL of the Jira server
  - The project that you selected
  - The status of the configuration
- Create defects for the test results that are available in your project in Rational® Test Automation Server. See [Creating Jira issues for tests that ran on page 538](#).

## Creating Jira issues for tests that ran

You can create Jira issues for the tests that ran in . The issues are created in the project in Jira that you configured in .

### Before you begin

- 
- Configured the Jira server as a change management system for your project in . See [Configuring the Jira server as a change management system on page 537](#).
- 
- 

### About this task


When you want to create a Jira issue in , the issue types that display in the **Create an issue in Jira** dialog depend on the configuration done by the administrator of the Jira server.


The default fields that display on the **Create an issue in Jira** dialog are as follows:

- **Issue Type**
- **Summary**
- **Description**

If you want to view any additional fields in the **Create an issue in Jira** dialog from , then you must inform your Jira server administrator to configure those additional fields as the required fields in the project on the Jira server. You can then use those fields in the **Create an issue in Jira** dialog when you create a defect from .

You can refer to the **Related information** section to know more about Jira field configuration.

- 1.
- 2.
- 3.
- 4.
5. Select the step that depends on the role assigned to you in the project on :
  - If you are a member with role assigned as *Owner*, go to [Step 6 on page 539](#).
  - If you are a member with role assigned as *Tester* and not completed the one-time authorization process, go to [Step 7 on page 539](#).
  - If you are a member with role assigned as *Tester* and completed the one-time authorization process, go to [Step 8 on page 540](#).
6. Perform the following steps as the project *Owner*:
  - a. Click the **Jira: Create a Jira issue** icon  that is under the **Actions** column in the row of the test.
 

Alternatively, you can click the test under the **Test** column to view the results panel, and then click the **Jira: Create a Jira issue** icon  that is displayed in the panel.
  - b. Select the issue type that you want to create in the **Create an issue in Jira** dialog, and then click **Create**.
 

The issue is created in the project on the Jira server and the link to the issue is displayed in the **Results** panel of the test on .

**Note:** You can create multiple issues for the same test result, if you want.

You created an issue in Jira. You can go to [Results on page 540](#).
7. Perform the following steps as a member assigned the *Tester* role and if you have not yet completed the one-time authorization process:



**Note:** You must complete the one-time authorization process if you are not a project owner.

- a.
- b. Click the URL of the Jira server in the **Configure Jira server to authorize the application** dialog that is displayed.
- c. Log in to the Jira server account with your valid credentials.



**Note:** The **Login** page is displayed only if you are not already logged in to the Jira server.

- d. Read the message that is displayed, and then click **Allow** to authorize the Jira server account.

After the authorization is successful, the **Access Approved** page is displayed with a verification code.

- e. Copy the verification code, and then enter the verification code in the field on the **Configure Jira server to authorize the application** dialog of .
- f. Click **Save**.
- g.

8. Perform the following steps as a member assigned the *Tester* role and completed the one-time authorization process:

- a.
- b.

## Results

You have created issues in the Jira project for the test results from .

The Jira issues are created in the Jira project and the links to the issues are displayed in the **Details** panel of the test result on the **Results** page.

## What to do next

You can click the test result on the **Results** page to view the Jira issue key link. You can then click the Jira issue key link to open the issue and view the details of the issue in Jira.

---

Related information

[Specifying Jira fields](#)



## Viewing details of the Jira issues

After you create Jira issues in Rational® Test Automation Server, you can view and track the details of the Jira issues in Jira.

### Before you begin

You must have completed the following tasks:

- Completed a test run and the test results are available on the **Results** page.
- Created Jira issues against the test results. See [Creating Jira issues for tests that ran on page 538](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Click **Active projects > My projects > project\_name** to open the project that contains the test assets.

The **Overview** page of the project is displayed.

3. Click **Analyze > Results**.

The **Results** page is displayed and all the test assets that were run earlier are displayed in tabs, as listed in the following table:

Tab	Description
<b>SUITES</b>	Lists all suites, compound tests, JMeter tests, JUnit tests, Postman tests, Rate Schedules, or VU Schedules that are in the selected branch of the repository.
<b>TESTS</b>	Lists all API tests, functional tests, or performance tests that are in the selected branch of the repository.
<b>ADVANCED</b>	Lists all assets that are displayed when custom filters are applied for assets in the selected branch of the repository.



**Note:** You can click the relevant tab to view the tests that were run earlier.

4. Click the test that is listed under the **Test** column.
5. Click the **Report** tab in the **Results** panel, if it is not already displayed.
6. Click the Jira issue link under **Defect tracking**.

The Jira log-in page is displayed, if you are not already logged into Jira.

7. Enter your username and password.
8. Click **Login**.

The issue details are displayed in Jira.



**Note:** The **Description** field of the Jira issue in Azure DevOps provides additional details of the test that ran in Rational® Test Automation Server such as the name of the test, the person who ran the test, the verdict of the test result, the date and time of the test run, the links to the test report that is generated in Rational® Test Automation Server, and the link to the query that is created.

## Results

You have viewed the details in Jira of the Jira issue that you created in Rational® Test Automation Server.

## Managing the Jira server configuration

You can manage the Jira server configuration that you set in Rational® Test Automation Server when you want to modify the project configured in Jira or when you want to delete the existing Jira configuration.

### Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role of a *Project Owner* in the project. See [Managing access to server projects on page 627](#).
- Ensured that you obtained the access and authentication details to Jira from the Jira administrator.

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.


2. Click **Active projects > My projects > *project\_name*** to open the project that contains the test assets.

The **Overview** page of the project is displayed.

3. Click **Manage > Integration** in the navigation pane.

The **Integration** page is displayed.

4. Click the **CHANGE MANAGEMENT SYSTEM** tab.

5. Click the **Open action menu** icon  for the Jira server that you want to manage.

The **Configure** and **Delete** options are displayed in the list.

6. Perform any of the following actions when you want to modify or delete the Jira server configuration:
  - To modify the project in Jira that was set previously in the Jira server configuration. Go to [Step 7 on page 542](#).
  - To delete the Jira server configuration. Go to [Step 8 on page 543](#).
7. Perform the following steps to modify the project in Jira:

- a. Select **Configure** as the option from the list.

The **Select project in Jira** dialog is displayed.

- b. Select the project that you want from the list displayed in the **Project** field.
  - c. Click **Update** to save the updated project in the Jira server configuration.
8. Perform the following steps to delete the existing Jira server configuration:
- a. Select **Delete** as the option from the list.  
The **Delete Jira configuration** dialog is displayed.
  - b. Click **Delete**.  
The Jira server configuration is deleted and a confirmation message is displayed.



**Note:** The links to the issues that you created for the tests that ran are not deleted from the results panel in Rational® Test Automation Server even after you delete the Jira server configuration.

## Results

You achieved the following results:

- Updated the project in Jira and saved the Jira server configuration.
- Deleted the Jira server configuration.

## Integrations with other applications

You can integrate certain applications with IBM® Rational® Test Automation Server to run tests, view the test results, and create defects.

### Integration plug-in compatibility matrix

You can find information about the versions of the integration plug-in that are compatible with IBM® Rational® Test Automation Server.

The following table lists the versions of the integration plug-in that are required to integrate Rational® Test Automation Server with IBM® Engineering Test Management, Jenkins, and UrbanCode™ Deploy.



**Note:** You must download the required version of the integration plug-in from the [IBM WebSphere, Liberty & DevOps Community](#) portal based on the existing version of Rational® Test Automation Server. You can then



integrate Rational® Test Automation Server with Engineering Test Management, Jenkins, and UrbanCode™ Deploy.

Rational® Test Automation Server	Engineering Test Management plug-in	Jenkins plug-in	UrbanCode™ Deploy plug-in
10.1.0	NA	RTAS-Jenkins-2.0	RTAS-UCD-1.0
10.1.1	NA	RTAS-Jenkins-3.0	RTAS-UCD-1.0
10.1.2	NA	RTAS-Jenkins-4.0	RTAS-UCD-2.0
10.1.3	NA	RTAS-Jenkins-4.0	RTAS-UCD-2.0
10.2.0	NA	RTAS-Jenkins-5.0	RTAS-UCD-3.0
10.2.1	IBM_RTAS_v10.2.1_ETM-ADAPTER_1.0.0	RTAS-Jenkins-5.0	RTAS-UCD-3.0
10.2.2	IBM_RTAS_v10.2.1_ETM-ADAPTER_1.0.0	RTAS-Jenkins-6.0	RTAS-UCD-3.0
10.2.3	IBM_RTAS_v10.2.3-ETM_ADAPTER_1.0.0-.v20220521_2256	RTAS-Jenkins-6.1	RTAS-UCD-3.2
10.5.0 10.5.1	IBM_RTAS_v10.5.0-ETM_ADAPTER_1.0.0-.v20220906_1035	RTAS-Jenkins-6.3	RTAS-UCD-3.3

## Integration with Azure DevOps

When you use Azure DevOps for continuous integration and continuous development of your application, you can run tests created for your application and available in a project on IBM® Rational® Test Automation Server, in Azure DevOps pipelines by using the *IBM Rational Test Workbench* extension.

### Overview

You can use the *IBM Rational Test Workbench* extension to integrate Rational® Test Automation Server with Azure DevOps.

The *IBM Rational Test Workbench* extension enables you to select any type of test available for your project in Rational® Test Automation Server that you can add to your task for the job in the Azure DevOps pipelines.

You can now follow the tasks listed in the task flow table to integrate Rational® Test Automation Server with Azure DevOps. See [Task flow for integrating Azure DevOps on page 545](#).

## Task flow for integrating IBM® Rational® Test Automation Server with Azure DevOps

The table shows the task flow for integrating Rational® Test Automation Server with Azure DevOps by using the *IBM Rational Test Workbench* extension. You must perform these tasks in sequence as listed in the following table. The table also provides you the links to the information about the tasks.

	Tasks	More information
1	Install Rational® Test Automation Server.	<a href="#">Installation of the server software on page 29</a>
2	Create tests in the desktop clients for the application that you want to test.	<ul style="list-style-type: none"> <li>• <a href="#">Rational® Performance Tester documentation</a></li> <li>• <a href="#">Rational® Functional Tester documentation</a></li> <li>• <a href="#">Rational® Test Workbench documentation</a></li> </ul>
3	Create a project on Rational® Test Automation Server and add the test assets that are created in the desktop clients to your project.	<a href="#">Test assets and a server project on page 616</a>
4	Create an organization and a project in Azure DevOps for running jobs in Azure DevOps pipelines.	<a href="#">Creating an organization</a>
5	Access the <b>Visual Studio Marketplace</b> portal and search for the latest version of the <i>IBM Rational Test Workbench</i> extension.	<a href="#">Visual Studio Marketplace</a>
6	Install the latest version of the <i>IBM Rational Test Workbench</i> extension.	<a href="#">Installing the IBM Rational Test Workbench extension on page 546</a>
7	Extend the trust certificate if you have used an internal CA certificate.	<a href="#">Certificate authority: Importing and extending lists on page 115</a>
8	Run tests in an Azure DevOps pipeline.	<a href="#">Running tests in an Azure DevOps Pipeline on page 547</a>

Related information

[Integration with Azure DevOps on page 544](#)

## Installing the *IBM Rational Test Workbench* extension

You must install the latest version of the *IBM Rational Test Workbench* extension in your Azure DevOps organization before you can use the extension for running your application tests in an Azure DevOps pipeline.

### Before you begin

You must have access to the **Visual Studio Marketplace** portal to install the latest version of the *IBM Rational Test Workbench* extension.

### About this task

After you install the latest version of the *IBM Rational Test Workbench* extension from the **Visual Studio Marketplace** portal in your Azure DevOps organization, you can select the tests that you want to run for your application in an Azure DevOps pipeline by using the *IBM Rational Test Workbench* extension.

1. Log in to the **Visual Studio Marketplace** portal, if you are not already logged in.
2. Click the **Azure DevOps** tab.
3. Search for the *IBM Rational Test Workbench* extension.
4. Click the *IBM Rational Test Workbench* extension.
5. Click **Get it free**.

#### Result

The **Visual Studio Marketplace** portal for the *IBM Rational Test Workbench* extension is displayed.

6. Select the organization where you want to run your test from the **Select an Azure DevOps Organization** list.
7. Click **Install**.

#### Result

The installation is completed.

8. Click **Proceed to organization**.

#### Result

The **Organization** page in Azure DevOps is displayed.

9. Click **Organization settings > Extensions**.

#### Result

The *IBM Rational Test Workbench* extension is displayed as an installed extension.

### Results

You have installed the *IBM Rational Test Workbench* extension in your Azure DevOps organization.

### What to do next

You can run tests that are available in Rational® Test Automation Server as a job in an Azure DevOps pipeline. See [Running tests in an Azure DevOps Pipeline on page 547](#).

## Running IBM® Rational® Test Automation Server tests in an Azure DevOps Pipeline

After you install the *IBM Rational Test Workbench* extension in your organization, you can run tests that are available in IBM® Rational® Test Automation Server as a job in Azure DevOps pipelines.

### Before you begin

You must have completed the following tasks:

- Added the tests that you created in the desktop clients for your application to the project on Rational® Test Automation Server.
- Installed the latest version of the *IBM Rational Test Workbench* extension in your organization. See [Installing the IBM Rational Test Workbench extension on page 546](#).
- Installed an agent in your pipeline. See [Azure Pipelines agents](#).
- Extended the CA list. See [Extending the trusted CA list on Linux on page 117](#) or [Extending the trusted CA list on Windows systems on page 118](#).
- Generated an offline user token from Rational® Test Automation Server. See [Generating an offline token on page 600](#).
- Created test cases under test plans if you want to view results of the test runs on the Test Plans dashboard.
- Created encrypted secrets or environment variables for your personal access token (PAT), if you want to use an encrypted PAT.

### About this task

After you add the *IBM Rational Test Workbench* extension in your Azure DevOps organization, you can use an existing pipeline or create a new one to add Rational® Test Automation Server test tasks. You can install an agent or use the one that you installed in your default agent pool. You can add Rational® Test Automation Server tests as tasks to your agent job, configure the task, and then run the task in the Azure DevOps pipeline.

If you have created test cases under test plans in a project on Azure DevOps, you must provide the details of the Azure DevOps URL, test plan, test case, and your PAT while you configure the test job in a pipeline so that you can view the results of the test run on your Test Plan dashboard.

1. Open your **Organization** page in Azure DevOps and perform the following steps:
  - a. Click the project you want to use.
  - b. Initialize the repository by performing the following steps:

- i. Click **Repos** from the left pane.
- ii. Click **Initialize** from the **Initialize with a README or gitignore** section.



**Note:** Select the **Add a README** checkbox if it is not selected.

- c. Click **Pipelines** from the left pane.
- d. Click **Create Pipeline**.
- e. Click **Use the classic editor** to create a pipeline without YAML.
- f. Verify the project, repository, and branch for manual and scheduled builds, and then click **Continue**.
- g. Click **Empty job**.


2. Select **Pipeline** and complete the following steps:

- a. Change the name for the build pipeline if required.
- b. Select the **Agent pool** for your build pipeline.

You can use the agent from the default agent pool or use the one you have installed.

- c. Select the **Agent Specification** for the agent if required.

3. Add a task to the agent job by completing the following steps:

- a. Click the **Add Task** icon  for the agent job.

**Result**

The **Add tasks** pane is displayed.

- b. Search for the IBM tasks defined in the *IBM Rational Test Workbench* extension.

**Result**

The tasks that you can select are displayed.



- c. Select the IBM® Rational® Test Automation Server Task, and then click **Add** to add the task to the agent job.



## Result

The selected task is added to the agent job and it is displayed with a warning that some settings require attention. You must configure the settings mentioned in [step 4 on page 549](#).

You can also remove the tasks that are not required in your job. Select the tasks in the list that you want to remove. You can then right-click the tasks, and click **Remove selected task(s)** to remove them.





4. Configure the settings by performing the following steps:



- a. Select the task version from the list if required.
- b. Follow the action for the task by referring to the following table:








**Note:** All mandatory fields are marked with an asterisk (\*) in the UI.

Field	Description	Action
Display name	The default name of the task. The default name that is displayed is IBM Rational Test Automation Server Task.	Enter a name for the task that helps you to identify the task by changing the default name.
IBM® Rational® Test Automation Server service connection	The details of IBM® Rational® Test Automation Server to set up a connection with Azure DevOps.	<p>Select the service connection from the drop-down list.</p> <p>If you are selecting the IBM® Rational® Test Automation Server service connection for the first time, you must click <b>New</b> to add the following details to add an IBM® Rational® Test Automation Server service connection, and then save the connection details:</p> <ul style="list-style-type: none"> <li>▪ <b>Server URL</b> - Enter the URL of Rational® Test Automation Server. The format of the URL is as follows: <code>https://hostname.</code></li> <li>▪ <b>Offline Token</b> - Enter the offline token that you generated from Rational® Test Automation Server.</li> <li>▪ <b>Service connection name</b> - Enter a service connection name.</li> </ul>

Field	Description	Action
		<ul style="list-style-type: none"> <li>▪ <b>Description (optional)</b> - Enter the details of the service connection if required.</li> <li>▪ Optionally, select the <b>Grant access permission to all pipelines</b> checkbox if required.</li> </ul> <p>You can save the service connection details. The service connection is available for selection from the IBM® Rational® Test Automation Server service connection drop-down list.</p> <p> <b>Note:</b></p> <p>You can edit or delete the service connection that you added if required. Click <b>Manage</b>, and then select the service connection from the <b>Service connections</b> list to open the service connection. You can click <b>Edit</b> to edit the service connection details. If you want to delete an existing service connection details, click the <b>Vertical ellipsis</b> icon, and then click <b>Delete</b> to delete the service connection details.</p> <p>Optionally, you can also create, edit, and delete a service connection from your Azure DevOps project dashboard. Open your project, click <b>Project settings &gt; Service connections</b>, and then perform any of the following tasks:</p> <ul style="list-style-type: none"> <li>▪ Create a new service connection.</li> <li>▪ Edit or delete an existing service connection that you already added.</li> </ul>
Team Space Name	The name of the team space on Rational® Test Automation Server that contains your project with the test assets.	<p>Enter the name of the team space that contains the project.</p> <p> <b>Note:</b> The license on the team space must be configured and you must be a member of that team space.</p>
Project	The name of the project containing the test assets in your team space on Rational® Test Automation Server.	<p>Enter the name of the project from the available names in the team space.</p> <p> <b>Note:</b></p> <p> <b>Note:</b> The <b>Project</b> drop-down list displays the projects where you are an <i>Owner</i> or a <i>member</i></p>

Field	Description	Action
		 of Rational® Test Automation Server. You must be an <i>Owner</i> or a <i>Tester</i> of the project that is available in the team spaces to run the tests from the Jenkins server.
		Azure DevOps pipeline.
Branch	The name of the branch of the repository in your project on Rational® Test Automation Server that contains the test assets.	Enter the branch where the test assets are stored.
Repository Link	The URL or link to the repository in your project that contains the test assets.	Enter the repository path that is configured for the project that contains the test to run.
File Path	The path of the test in the repository of your project.	Enter the path of the test in Rational® Test Automation Server that you want to run.
API Test Environment	The environment defined in the API Suite or test to use in the test run.	Enter the name of the environment that is configured in the test asset for the test created in Rational® Integration Tester.   <b>Note:</b> This field is applicable only if you are running an API suite test.
Variables	The variable name and its value.	Enter the name of the variable and its value if your test requires variables during the test run time.  You must enter the variables in the following format:  <code>name_of_the_variable=value_of_the_varibale</code>  You can add multiple variables to the test run separated by a semicolon.  For example, <code>varname1=value1;varname2=value2</code>
Datasets	The name with path to the dataset that must be used during the test run.	Enter the path to the dataset if you want to replace the values of the dataset during a test run.

Field	Description	Action
		<p>You must ensure that both original and new datasets are in the same workspace and have the same column names. When you enter a value for the <b>Datasets</b> field, you must also include the path to the dataset. The path must be in the following format:</p> <pre>/project_name/ds_path/original_datasets.csv:/project_name/ds_path/new_datasets.csv</pre> <p> <b>Note:</b> You can override multiple datasets that are saved in a different project by adding multiple paths to the dataset separated by a semicolon.</p>
Labels	The option to add labels to the test results when the test run is complete.	<p>Enter the labels to add to test results when the test run is complete.</p> <p>You can add multiple labels to a test result separated by a comma.</p> <p>For example, <i>label1, label2</i></p> <p>After the test run is complete, then the values provided in the <b>Labels</b> field are displayed on the <b>Results</b> page of Rational® Test Automation Server.</p>
Secrets Collection Name	The name of the secrets collection that contains the secrets to be used in certain API Suites or tests.	<p>Enter the name of the secrets collection that contains the secrets to be used in the API Suite or test.</p> <p> <b>Note:</b> This field is applicable only if you are running an API suite test.</p>
Azure DevOps Project URL	The URL of the test project in the organization on the Azure DevOps server.	<p>Enter the URL of the test project in the organization on the Azure DevOps server in the following format:</p> <pre>https://&lt;host&gt;/&lt;orgname&gt;/&lt;projectName&gt;</pre> <p>You must use this option with the following options:</p> <ul style="list-style-type: none"> <li>▪ Azure DevOps PAT</li> <li>▪ Test Plan Name</li> <li>▪ Test Case Name</li> </ul> <p> <b>Note:</b> You must enter the details for this option if you have created</p>

Field	Description	Action
	 test cases under test plans in your Azure project and want to view the test results on the Test Plans Dashboard.	
Azure DevOps PAT	The personal access token (PAT) of the Azure server where the test project that contains the test plan is hosted. You can also enter the variable for the encrypted PAT that you create in Azure DevOps.	<p>Enter your personal access token.</p> <p>You must use this option with the following options:</p> <ul style="list-style-type: none"> <li>▪ Azure DevOps Project URL</li> <li>▪ Test Plan Name</li> <li>▪ Test Case Name</li> </ul>
	 <b>Note:</b> The token must have read and write access.	
Test Plan Name	The name of the test plan in your Azure project.	<p>Enter the name of the test plan.</p> <p>You must use this option with the following options:</p> <ul style="list-style-type: none"> <li>▪ Azure DevOps Project URL</li> <li>▪ Azure DevOps PAT</li> <li>▪ Test Case Name</li> </ul>
Test Case Name	The name of the test case under the test plan in your Azure project.	<p>Enter the name of the test case.</p> <p>You must use this option with the following options:</p> <ul style="list-style-type: none"> <li>▪ Azure DevOps Project URL</li> <li>▪ Azure DevOps PAT</li> <li>▪ Test Plan Name</li> </ul>
Export JUnit Report	The location to store the JUnit report of the test.	<p>Enter the path to store the JUnit report that is generated for the test in the XML format.</p> <p>You must specify the complete path to the directory including the file name of the report.</p>

Field	Description	Action
-------	-------------	--------

For example, `C:/Users/Documents/Reports/junit_test_report.xml`

- c. Expand **Control Options** and configure the settings for your task if required.
  - d. Expand **Output Variables** and configure the settings for your task if required.
5. Select from the following options:
- a. Click **Save** to save the configured settings for the task.



**Note:** The task is not queued for a run.

You can save the task to a build pipeline and opt to run the build at a later time.

- b. Click **Save & queue** to save the configurations and queue the run in the pipeline.

**Result**

The **Run pipeline** dialog is displayed.

6. Complete the following steps:
- a. Enter a comment for the test in the **Save comment** field.
  - b. Select the agent that you configured for the test from the **Agent pool** list.
  - c. Select the agent specification from the **Agent Specification** list if required.
  - d. Select the branch from the **Branch/tag** list.
  - e. Add the variables and demands for the task run from the **Advanced Options** pane if required.
  - f. Select the **Enable system diagnostics** checkbox for a detailed log view.
  - g. Click **Save and run**.

**Result**

The `pipeline summary` page displays the progress of the job run.

**Results**

You have run the tests for the application that you are testing in the Azure DevOps pipeline.

**What to do next**

You can perform any of the following actions in Azure DevOps:

- View the test results on the **Task** page by clicking the task.

The **Reports information** section on the **Task** page displays the names of the report along with its corresponding URLs. The report URLs are the Rational® Test Automation Server URLs where the reports are stored. You can access the report URLs to view the test run information at any point of time.

- View the task logs from the **pipeline summary** page for the job you ran.
- View the test results on the Test Plans Dashboard if you created test cases under test plans in your Azure project.

You can perform any of the following actions in Rational® Test Automation Server:

- View the test results of the test that you ran from Azure DevOps on the **Results** page.
- View the logs and reports of the test that you ran from Azure DevOps by clicking the result panel of the test asset or resource, and then click the log or the report you want to view.

## Integration with IBM® Engineering Test Management

As a *Project Owner*, you can integrate IBM® Engineering Test Management as a quality management solution with IBM® Rational® Test Automation Server. You can run test assets that are available in a project from Engineering Test Management.

The following topics provides more information about the integration of IBM® Rational® Test Automation Server with Engineering Test Management.

### Task flow for integration of IBM® Engineering Test Management

You must perform the following tasks in sequence to integrate IBM® Rational® Test Automation Server with IBM® Engineering Test Management. For more information about the tasks, refer to the links provided in the table.

Sl. No	Tasks	More information
1	Create a project in Rational® Test Automation Server.	<a href="#">Adding a project on page 619</a>
2	Install the required version of Engineering Test Management on your computer.	<a href="#">Engineering Lifecycle Management Documentation</a>
3	Download the Engineering Test Management adapter, and then extract the contents of a compressed file.  For more information about specific versions of plug-in, see <a href="#">Integration plug-in compatibility matrix on page 543</a> .	<a href="#">IBM WebSphere, Liberty &amp; DevOps Community portal</a>

Sl. No	Tasks	More information
4	Add the adapter files that you downloaded to the installation directory of Engineering Test Management.	<a href="#">Adding the adapter files to IBM Engineering Test Management on page 556</a>
5	Configure the details of Engineering Test Management on Rational® Test Automation Server.	<a href="#">Configuring IBM Engineering Test Management on Rational Test Automation Server on page 557</a>
6	Create a test script and a test case on Engineering Test Management to run the test assets.	<a href="#">Preparing test cases in IBM Engineering Test Management on page 559</a>
7	Run the test assets from Engineering Test Management.	<a href="#">Running tests from IBM Engineering Test Management on page 560</a>

## Adding the adapter files to IBM® Engineering Test Management

You must add the adapter files to the installation directory to run test assets of Rational® Test Automation Server from IBM® Engineering Test Management.

### Before you begin

You must have completed the following tasks:

- Installed the latest version of Engineering Test Management from the [Jazz](#) portal on your computer. See [System Requirements on page 12](#).
- Downloaded the adapter (a compressed file) for Engineering Test Management from the [IBM WebSphere, Liberty & DevOps Community](#) portal .

For more information about specific versions of plug-in, see [Integration plug-in compatibility matrix on page 543](#).

### About this task

You must add the adapter files to the location where you installed Engineering Test Management to run test assets of Rational® Test Automation Server.

1. Extract the compressed file that you downloaded from the Test Workbench Resource Community portal.
2. Copy the `etm-adapter-updatesite` folder.
3. Go to the location where you installed Engineering Test Management on your computer.
4. Place the `etm-adapter-updatesite` folder in the `sites` folder of the installation directory of Engineering Test Management.

For example, `D:\ELM\JazzTeamServer\server\conf\qm\sites`.

5. Copy the `rtas-profile.ini` file from the `provision_profiles` folder that is available in the extracted zip file.



- Place the `rtas-profile.ini` file in the `provision_profiles` folder of the installation directory of Engineering Test Management.

For example, `D:\ELM\JazzTeamServer\server\conf\qm\provision_profiles`

### Results

You have added the adapter files to Engineering Test Management.

### What to do next

You must perform the following tasks:

- You must reset the Engineering Test Management server from the web console by using the **Request Server Reset** option.

If you have administrator privileges, then you can access the **Request Server Reset** option from the following URL:

```
https://<hostname>:<portnumber>/qm/admin?
internal#action=com.ibm.team.repository.admin.serverReset
```



**Note:** You must replace the following values with the actual values in the URL:

- `<hostname>` with the hostname that you used when you installed Engineering Test Management.
- `<portnumber>` with the port number that is associated with an IP address of the hostname.
- Restart the Engineering Test Management server.
- Configure the details of Engineering Test Management on Rational® Test Automation Server to establish a connection between the two products. See [Configuring IBM Engineering Test Management on Rational Test Automation Server on page 557](#).

## Configuring IBM® Engineering Test Management on Rational® Test Automation Server

You must configure IBM® Engineering Test Management on Rational® Test Automation Server to establish a connection between the two products.

### Before you begin

You must have completed the following tasks:

- Granted sufficient privileges to access **Project area** on Engineering Test Management.
- Created a project area in Engineering Test Management.
- Created a project in Rational® Test Automation Server.

### About this task

If you have multiple instances of Engineering Test Management, then you can configure all those instances on Rational® Test Automation Server. If you have only one instance of Engineering Test Management, you can configure the same instance multiple times. However, you must choose a different **Project area** during the configuration of Engineering Test Management.

1. Log in to Rational® Test Automation Server, if you are not already logged in.
2. Open a project from the team space.
3. Click **Manage > Integration**.
4. Click the **TEST MANAGEMENT** tab, and then click **Add IBM ETM adapter**.
5. Enter the following information in the **Add IBM ETM adapter** dialog:

Fields	Actions
<b>Server URL</b>	Enter the URL of Engineering Test Management.  For example, <code>https://&lt;hostname&gt;:&lt;portnumber&gt;/qm</code>
<b>Username</b>	Enter the username associated with Engineering Test Management.
<b>Password</b>	Enter the password associated with the username of Engineering Test Management.

6. Click **Validate**.
7. Select the project name that you created in Engineering Test Management from the **Project area** list.
8. Enter a name for the Engineering Test Management adapter in the **Adapter name** field.



**Note:** You cannot provide an adapter name that already exists for the **Project area**.

9. Click **Save**.

### Results

You have configured the details of Engineering Test Management on Rational® Test Automation Server.

### What to do next

You must create a test script and test case on Engineering Test Management to run the test assets of Rational® Test Automation Server. See [Preparing test cases in IBM Engineering Test Management on page 559](#).

## Preparing test cases in IBM® Engineering Test Management



You must create a test script and a test case on IBM® Engineering Test Management to run the test assets.

### Before you begin

You must have completed the following tasks:

- Added the adapter files to the installation directory of Engineering Test Management. See [Adding the adapter files to IBM Engineering Test Management on page 556](#).
- Configured the details of Engineering Test Management on Rational® Test Automation Server. See [Configuring IBM Engineering Test Management on Rational Test Automation Server on page 557](#).
- Generated an offline user token from Rational® Test Automation Server. See [Generating an offline token on page 600](#).
- Copied the following values to be used during the preparation of a test script and a test case:
  - Branch name of the Git repository
  - Asset ID of the test that you want to run
  - **Optional:** Name of the environment if you want to run the API test suite.

1. Log in to Engineering Test Management, if you are not already logged in.
2. Open your project area from where you want to run the test assets.
3. Click **Construction**, and then click **Test Script** from the drop-down list.
4. Enter a name for the test script in the **Test Script** field.
5. Select **Rational Test Automation Server** from the **Type** drop-down list.
6. Enter the following test information:

Fields	Actions
Offline Token	Enter the offline token that you generated from Rational® Test Automation Server.
Branch	Enter the name of the branch where your test assets are stored.
Asset ID	Enter an asset ID of the test asset that is available on the <b>Execution</b> page of Rational® Test Automation Server.   <b>Tip:</b> You can hover over the name of the test asset to view and copy the asset ID.
Environment	Enter a name of the API test environment.   <b>Remember:</b> You must use this field only if you want to run the API test suites.

7. Click **Save**.

8. Perform the following steps to add a test script to a test case:

a. Click **Construction > Test Case**.

**Result**

The **Test Cases** page is displayed.

b. Enter a name for the test case in the **Test Case** field.

c. Click **Test Scripts** from the left navigation pane.

d. Click the **Plus** icon from the **Test Scripts** panel.

**Result**

The **Add Test Scripts** dialog is displayed.

e. Select the checkbox to add the test script that you created to the test case, and then click **Add and Close**.

f. Click **Save**.

**Results**

You have prepared Engineering Test Management.

**What to do next**

You can run the test assets of Rational® Test Automation Server from Engineering Test Management. See [Running tests from IBM Engineering Test Management on page 560](#).


## Running tests from IBM® Engineering Test Management

After you create a test case in IBM® Engineering Test Management, you can run the test. The test case includes the test scripts that contains test assets of Rational® Test Automation Server.

**Before you begin**


You must have performed the following tasks:

- Prepared Engineering Test Management to run test assets of Rational® Test Automation Server. See [Preparing test cases in IBM Engineering Test Management on page 559](#).
- Configured the details of Engineering Test Management on Rational® Test Automation Server. See [Configuring IBM Engineering Test Management on Rational Test Automation Server on page 557](#).

1. Log in to Engineering Test Management.
2. Open the test case that you created in Engineering Test Management.
3. Click the **Play** icon  , and then click **Run**.

**Result**

The **Run Test Case** dialog is displayed.

4. Click the **ChangeMachine** icon  next to the **Machine** field to select the adapter that you configured on Rational® Test Automation Server.
5. Select the required adapter name from the list, and then click **OK**.
6. Click **Finish** to start the test run.

The Engineering Test Management dashboard displays the progress of the test case run.

## Results

You have run the test assets of Rational® Test Automation Server from Engineering Test Management.

## What to do next

You can perform the following tasks:

- Click **Show Result** to view information about the test results from Engineering Test Management when the test run is complete.
- Click **Link to Result** from **Results details** to view the test results from the **Results** page of Rational® Test Automation Server. See [Test results and reports on page 456](#).

## Deleting an IBM® Engineering Test Management adapter


You might want to delete an IBM® Engineering Test Management adapter when it is no longer needed.

### Before you begin

You must have completed the following tasks:

- Been assigned the role of *Owner* of a project or *Owner* of the project in Rational® Test Automation Server.
  - Configured the details of Engineering Test Management on Rational® Test Automation Server. See [Configuring IBM Engineering Test Management on Rational Test Automation Server on page 557](#).
1. Log in to Rational® Test Automation Server and open the team space that contains your project.
  2. Click **Manage > Integration**.
  3. Click the **TEST MANAGEMENT** tab.

You can view the Engineering Test Management adapters that are configured with the project on Rational® Test Automation Server.

4. Identify the Engineering Test Management adapter that you want to delete, and then click the **Menu** icon .
5. Click **Delete**.



**Note:** If any of the test runs are in progress on Engineering Test Management, then those runs are aborted.

6. Click **Confirm** to delete the Engineering Test Management adapter from the project.

## Results

You have deleted the Engineering Test Management adapter from the project on Rational® Test Automation Server.

## Integration with Jenkins

You can use the IBM® Rational® Test Automation Server plug-in to integrate Rational® Test Automation Server with Jenkins. After you complete the integration, you can run tests assets that are available in a project of Rational® Test Automation Server from a Jenkins server.

You must have installed Jenkins on your computer to install the Rational® Test Automation Server Jenkins plug-in. You can run the following command to start the Jenkins server to support UTF-8 character sets:

```
java -Dfile.encoding=UTF8 -jar jenkins.war
```

You can use either the Jenkins **Freestyle** project or the **Pipeline** project to run test assets from Jenkins. With **Freestyle** project, you can create build step from the Jenkins UI to run the test assets. Whereas the **Pipeline** project uses a simple text scripts based on the Groovy programming language. You can define pipeline scripts during the configuration of the **Pipeline** project by using any of the following methods:

- **Pipeline script:** In this method, you can directly enter a script in the field provided within the Jenkins UI or you can select a sample **Pipeline script** from the drop-down list.
- **Pipeline script from SCM:** In this method, you can create a script with a Groovy editor and then commit the script file into the Git repository.

For more information about the Jenkins Pipeline, refer to the [Jenkins](#) documentation.

Refer to the following topics to learn more about integrating Jenkins with Rational® Test Automation Server.

## Task flows for running test assets from Jenkins

You can perform certain tasks to run test assets from the Jenkins **Freestyle** project or the **Pipeline** project.

The following table lists the task flows for running test assets from the Jenkins **Freestyle** project:

Tasks	Go to...
Install the Rational® Test Automation Server Jenkins plug-in.	<a href="#">Installing the Rational Test Automation Server Jenkins plug-in on page 563</a>
Import the CA certificate to your custom trust store.	<a href="#">Using a custom trust store on page 564</a>
Configure the <b>Freestyle</b> project.	<a href="#">Configuring the Freestyle project on page 566</a>
Run Rational® Test Automation Server test assets on Jenkins.	<a href="#">Running tests from Jenkins on page 576</a>

You can perform the following tasks to run test assets from the Jenkins **Pipeline** project by using the sample script:

Tasks	Go to...
Install the Rational® Test Automation Server Jenkins plug-in.	<a href="#">Installing the Rational Test Automation Server Jenkins plug-in on page 563</a>
Import the CA certificate to your custom trust store.	<a href="#">Using a custom trust store on page 564</a>
Create a pipeline script from the Jenkins UI.	<a href="#">Creating a pipeline script from Jenkins on page 570</a>
Configure the <b>Pipeline</b> project.	<a href="#">Configuring the Pipeline project by using the sample script on page 574</a>
Run Rational® Test Automation Server test assets on Jenkins.	<a href="#">Running tests from Jenkins on page 576</a>

You can perform the following tasks to run test assets from the Jenkins **Pipeline** project by using the script from Source Code Management (SCM):

Tasks	Go to...
Install the Rational® Test Automation Server Jenkins plug-in.	<a href="#">Installing the Rational Test Automation Server Jenkins plug-in on page 563</a>
Import the CA certificate to your custom trust store.	<a href="#">Using a custom trust store on page 564</a>
Create a pipeline script and commit the script to the Git repository.	For more information, refer to the <a href="#">Jenkins</a> documentation.
Configure the <b>Pipeline</b> project.	<a href="#">Configuring the Pipeline project by using the script from SCM on page 575</a>
Run Rational® Test Automation Server test assets on Jenkins.	<a href="#">Running tests from Jenkins on page 576</a>

## Installing the Rational® Test Automation Server Jenkins plug-in

You must install the Rational® Test Automation Server Jenkins plug-in on the Jenkins server before you can use the plug-in for running tests that are available in your Rational® Test Automation Server project on the Jenkins server.

### Before you begin

You must have completed the following tasks:

- Installed Jenkins on your computer. For more information about installing Jenkins, refer to the [Jenkins documentation](#).
- Downloaded the Rational® Test Automation Server Jenkins plug-in from the [IBM WebSphere, Liberty & DevOps Community portal](#).

For more information about specific versions of plug-in, see [Integration plug-in compatibility matrix on page 543](#).

1. Log in to the Jenkins server.

**Result**

The Jenkins dashboard is displayed.

2. Click **Manage Jenkins > Manage Plugins**.
3. Click the **Advanced** tab.
4. Click **Choose File** from the **Upload plug-in** section to locate and **Open** the Rational® Test Automation Server Jenkins plug-in.
5. Click **Upload**.

**Result**

The Rational® Test Automation Server Jenkins plug-in is displayed in the **Installed** tab.

**Results**

You have installed the Rational® Test Automation Server Jenkins plug-in on the Jenkins server.

**What to do next**

You must import the CA certificate to your custom trust store to establish a trusted and secure connection between the Jenkins server and Rational® Test Automation Server. See [Using a custom trust store on page 564](#).

## Using a custom trust store

You must use a custom trust store in the Jenkins build to establish a trusted and secure connection between the Jenkins server and Rational® Test Automation Server.

**Before you begin**

You must have completed the following tasks:

- Saved the Certificate Authority (CA) assigned to Rational® Test Automation Server.



**Note:** You can contact the administrator of Rational® Test Automation Server if you do not have a copy of the CA.

- Installed Java on the computer from where you access the Rational® Test Automation Server UI.

**About this task**



If the SSL certificate assigned to Rational® Test Automation Server is signed by an internal CA, then you must download and import the CA to a custom trust store. You can then use the custom trust store in the Jenkins build step to establish a trusted and secure connection between the Jenkins server and Rational® Test Automation Server.



**Note:** If the internal CA is already imported to the default trust store that is used by the Jenkins server, you need not use a custom trust store.



**Restriction:**

When you use Red Hat Enterprise Linux (RHEL) operating systems for Jenkins, you must run the Jenkins service with a user who has access to the custom trust store path to utilize the custom trust store feature. To change the Jenkins user, you must open the `/etc/sysconfig/jenkins` file and set the `JENKINS_USER` to the user who has access to the custom trust store path.

You can then run the following commands to set `JENKINS_USER` to another user who has access to the path of the custom trust store:

```
$JENKINS_USER= <username>
```

For example, `$JENKINS_USER= <user1>`



**Note:** You must ensure that the user account is available in the `/etc/passwd` file.

You can then run the following commands to change the ownership of the Jenkins folder:

```
chown -R username:username /var/lib/jenkins
```

```
chown -R username:username /var/cache/jenkins
```

```
chown -R username:username /var/log/jenkins
```

For example,

```
chown -R user1:user1 /var/lib/jenkins
```

```
chown -R user1:user1 /var/cache/jenkins
```

```
chown -R user1:user1 /var/log/jenkins
```

After the change of ownership is complete, you can run the following command to restart the Jenkins server:

```
/etc/init.d/jenkins restart
```

1. Locate the default trust store file (`cacerts` file) from the JRE directory.  
For example: `C:\Program Files\Java\jdk-18.0.1.1\lib\security`
2. Copy the `cacerts` file to a location of your choice on your computer.

- Place the CA assigned to Rational® Test Automation Server in the same location where you saved the `cacerts` file.
- Run the following command from the command-line interface or terminal to import the CA of Rational® Test Automation Server to custom trust store:

```
keytool -import -trustcacerts -file {path to the downloaded CA with the file extension} -alias {label for the certificate} -keystore {path to the trust store cacerts file}
```

For example,

```
keytool -import -trustcacerts -file C:\Users\Desktop\cert\ca-master-ibm.crt -alias "rtasca" -keystore C:\Users\Desktop\cert\cacerts
```



**Note:** The default password of the trust store is *changeit*. It remains the same for the custom trust store. If you want to change the password, you can run the following command, and then enter the new password:

```
keytool -storepasswd -keystore {path to the trust store cacerts file}
```

For example, `keytool -storepasswd -keystore C:\Users\Desktop\cert\cacerts`

## Results

You have successfully imported the downloaded CA to the custom trust store.

## What to do next

You can run test assets either from **Freestyle** or **Pipeline** project from the Jenkins server.

## Configuring the Freestyle project

You must configure a **Freestyle** project to add a build step, and then run tests that are available in your Rational® Test Automation Server project from Jenkins.

### Before you begin

You must have completed the following tasks:

- Installed the Rational® Test Automation Server Jenkins plugin on the Jenkins server. See [Installing the Rational Test Automation Server Jenkins plug-in on page 563](#).
- Created a Jenkins **Freestyle** project.
- Generated an offline user token from Rational® Test Automation Server. See [Managing access to IBM Rational Test Automation Server on page 600](#).


- Open your Jenkins **Freestyle** project from the list, and then click **Configure**.
- Click the **Build** tab, and then click **Add build step**.
- Select the **Run IBM Rational Test Automation Server test** option from the drop-down list.



4. Provide the details about the test run for the fields in the following table:



The following table lists the required fields that you must provide to run the test from Jenkins:

Fields	Action
<b>Name</b>	Enter a name for the Jenkins build step.
<b>Server URL</b>	Enter the URL of Rational® Test Automation Server.  The format of the URL is as follows: <i>https://hostname</i> .
<b>Offline Token</b>	Enter the offline user token that you generated from Rational® Test Automation Server.

The following table lists the optional fields that you can provide to run the test from Jenkins:

Fields	Descriptions
<b>Asset Type</b>	Use this field to select the test asset type that you want to run from the <b>Asset Type</b> drop-down list.
<b>Branch</b>	Use this field to select the branch of the Git repository from the drop-down list.  The <b>Branch</b> drop-down list displays the branches that are available in the corresponding project of Rational® Test Automation Server.   <b>Note:</b> After you select the branch from the <b>Branch</b> drop-down list, if you want to change the URL of Rational® Test Automation Server, offline user token, or the project in the <b>Build</b> step, then you must close the build step. Then you must click <b>Add build step</b> , select <b>Run Rational Test Automation Server test</b> , and then enter the details.
<b>Datasets</b>	Use this field to enter the path to the dataset if you want to replace the values of the dataset during a test run.  You must ensure that both original and new datasets are in the same workspace and have the same column names. When you enter a value for the <b>Datasets</b> field, you must also include the path to the dataset. The path must be in the following format:  <code>/project_name/ds_path/original_ds.csv:/project_name/ds_path/new_ds.csv</code>

Fields	Descriptions
	 <b>Note:</b> You can override multiple datasets that are saved in a different project by adding multiple paths to the dataset separated by a semicolon.
<b>Labels</b>	<p>Use this field to add labels to test results when the test run is complete.</p> <p>You can add multiple labels to a test result separated by a comma.</p> <p>For example, <i>label1, label2</i></p> <p>After the test run is complete, then the values provided in the <b>Labels</b> field are displayed on the <b>Results</b> page of Rational® Test Automation Server.</p>
<b>Project</b>	<p>Use this field to select a project from the drop-down list.</p> <p>The <b>Project</b> drop-down list displays the projects that are available in the corresponding team space of Rational® Test Automation Server.</p>  <b>Note:</b> The <b>Project</b> drop-down list displays the projects where you are an <i>Owner</i> or a <i>member</i> of Rational® Test Automation Server. You must be an <i>Owner</i> or a <i>Tester</i> of the project that is available in the team spaces to run the tests from the Jenkins server.
<b>Secrets Collection</b>	<p>Use this field to select the Secret from the <b>Secrets Collection</b> drop-down list if you created secrets collections for your project to use Secrets at test run time.</p> <p>The <b>Secrets collection</b> field is mandatory only if you select one of the following options in the <b>Asset Type</b> field:</p> <ul style="list-style-type: none"> <li>◦ <b>APISUITE</b></li> <li>◦ <b>APITEST</b></li> <li>◦ <b>APISTUB</b></li> <li>◦ <b>POSTMAN</b></li> </ul> <p>The following message is displayed when you select any other asset type than preceding list:</p> <pre>You can select Secrets collection only if you are running tests of type APISUITE, APITEST, APISTUB, or POSTMAN.</pre>
<b>Team Space</b>	<p>Use this field to select the name of the team space from the drop-down list.</p> <p>The <b>Team Space</b> drop-down list displays the names of the team spaces if you are an <i>Owner</i> or a <i>member</i> of the team space.</p>

Fields	Descriptions
<b>Test</b>	<p>Use this field to select the required test that you want to run from the drop-down list.</p> <p>The <b>Tests</b> drop-down list displays the available test assets from the corresponding branch in the following format:</p> <pre>Project_name [Path: the path of the test assets] [Repo: URL of the Git repository that the test belongs to]</pre> <p> <b>Note:</b> After you select the test details from the <b>Tests</b> drop-down list if you want to change the URL of Rational® Test Automation Server, offline user token, or the project in the Jenkins <b>Build step</b>, then you must delete the build step and add a new build step.</p>
<b>Test Environment</b>	<p>Use this field to select an environment from the drop-down list.</p> <p>The <b>Test Environment</b> field displays the available test environments from Rational® Test Automation Server.</p> <p>The <b>Test Environment</b> field is mandatory only if you select one of the following options in the <b>Asset Type</b> field:</p> <ul style="list-style-type: none"> <li>◦ <b>APISUITE</b></li> <li>◦ <b>APITEST</b></li> <li>◦ <b>APISTUB</b></li> <li>◦ <b>POSTMAN</b></li> </ul> <p>The following message is displayed when you select any other asset type than preceding list:</p> <pre>You can select Test Environment only if you are running tests of type APISUITE, APITEST, APISTUB, or POSTMAN.</pre>
<b>Use Custom Trust Store</b>	<p>Select this checkbox if you used an internal CA certificate and imported the certificate to the custom trust store. You can then enter the file path of your trust store followed by the file name in the <b>Custom Trust Store Path</b> field.</p> <p>Select the <b>Use Custom Password for the Trust Store</b> checkbox if you modified the trust store password, and then enter the new password in the <b>Custom Trust Store Password</b> field.</p> <p> <b>Note:</b> If you did not modify the trust store password, you must clear the <b>Use Custom Password for the Trust Store</b> checkbox.</p>
<b>Variables</b>	<p>Use this field to enter the name of the variable and its value if your test requires variables during the test run time.</p>

Fields	Descriptions
	<p>You must enter the variables in the following format:</p> <pre>name_of_the_variable=value_of_the_varibale</pre> <p>You can add multiple variables to the test run separated by a semicolon.</p> <p>For example, <i>varname1=value1;varname2=value2</i></p>



**Note:** You can run multiple tests sequentially in the same job by adding multiple build steps and providing details for the tests that you want to run.

5. Click **Save** to save the build step.

### Results

You have configured the **Freestyle** project by adding the build step.

### What to do next

You can run test assets from the Jenkins server. See [Running tests from Jenkins on page 576](#).

## Creating a pipeline script from Jenkins

You must create a pipeline script to run test assets that are available in the Rational® Test Automation Server project from the Jenkins **Pipeline** project.

### Before you begin


You must have completed the following tasks:


- Installed the Rational® Test Automation Server Jenkins plugin on the Jenkins server. See [Installing the Rational Test Automation Server Jenkins plug-in on page 563](#).
- Generated an offline user token from Rational® Test Automation Server. See [Managing access to IBM Rational Test Automation Server on page 600](#).
- Created a Jenkins **Pipeline** project.

1. Open your Jenkins **Pipeline** project from the list.
2. Click **Configure**, and then select the **Pipeline** tab.
3. Click **Pipeline Syntax** to generate a pipeline script.
4. Select the **step: General Build Step** option from the drop-down list in the **Sample Step** field.
5. Select the **Run IBM Rational Test Automation Server test** option from the drop-down list in the **Build Step** field.
6. Provide the details about the test run for the fields in the following table:



The following table lists the required fields that you must provide to run the test from Jenkins:

Fields	Action
<b>Name</b>	Enter a name for the Jenkins build step.
<b>Server URL</b>	Enter the URL of Rational® Test Automation Server.  The format of the URL is as follows: <i>https://hostname</i> .
<b>Offline Token</b>	Enter the offline user token that you generated from Rational® Test Automation Server.

Field	Description
<b>Asset Type</b>	Use this field to select the test asset type that you want to run from the <b>Asset Type</b> drop-down list.
<b>Branch</b>	Select the branch from the drop-down list.  The <b>Branch</b> drop-down list displays the branches that are available in the corresponding project of Rational® Test Automation Server.
<b>Datasets</b>	Use this field to enter the path to the dataset if you want to replace the values of the dataset during a test run.  You must ensure that both original and new datasets are in the same workspace and have the same column names. When you enter a value for the <b>Datasets</b> field, you must also include the path to the dataset. The path must be in the following format:  <code>/project_name/ds_path/original_ds.csv:/project_name/ds_path/new_ds.csv</code>   <b>Note:</b> You can override multiple datasets that are saved in a different project by adding multiple paths to the dataset separated by a semicolon.
<b>Labels</b>	Use this field to add labels to test results when the test run is complete.  You can add multiple labels to a test result separated by a comma.  For example, <i>label1, label2</i>

Field	Description
	After the test run is complete, then the values provided in the <b>Labels</b> field are displayed on the <b>Results</b> page of Rational® Test Automation Server.
<b>Project</b>	<p>Use this field to select a project from the drop-down list.</p> <p>The <b>Project</b> drop-down list displays the projects that are available in the corresponding team space of Rational® Test Automation Server.</p> <p> <b>Note:</b> The <b>Project</b> drop-down list displays the projects where you are an <i>Owner</i> or a <i>member</i> of Rational® Test Automation Server. You must be an <i>Owner</i> or a <i>Tester</i> of the project that is available in the team spaces to run the tests from the Jenkins server.</p>
<b>Secrets Collection</b>	<p>Use this field to select the Secret from the <b>Secrets Collection</b> drop-down list if you created secrets collections for your project to use Secrets at test run time.</p> <p>The <b>Secrets collection</b> field is mandatory only if you select one of the following options in the <b>Asset Type</b> field:</p> <ul style="list-style-type: none"> <li>◦ <b>APISUITE</b></li> <li>◦ <b>APITEST</b></li> <li>◦ <b>APISTUB</b></li> <li>◦ <b>POSTMAN</b></li> </ul> <p>The following message is displayed when you select any other asset type than preceding list:</p> <p>You can select Secrets collection only if you are running tests of type APISUITE, APITEST, APISTUB, or POSTMAN.</p>
<b>Team Space</b>	<p>Use this field to select the name of the team space from the drop-down list.</p> <p>The <b>Team Space</b> drop-down list displays the names of the team spaces if you are an <i>Owner</i> or a <i>member</i> of the team space.</p>
<b>Test</b>	<p>Use this field to select the required test that you want to run from the drop-down list.</p> <p>The <b>Tests</b> drop-down list displays the available test assets from the corresponding branch in the following format:</p> <p>Project_name [Path: the path of the test assets] [Repo: URL of the Git repository that the test belongs to]</p>



Field	Description
	 <b>Note:</b> After you select the test details from the <b>Tests</b> drop-down list if you want to change the URL of Rational® Test Automation Server, offline user token, or the project in the Jenkins <b>Build step</b> , then you must delete the build step and add a new build step.
<b>Test Environment</b>	<p>Use this field to select an environment from the drop-down list.</p> <p>The <b>Test Environment</b> field displays the available test environments from Rational® Test Automation Server.</p> <p>The <b>Test Environment</b> field is mandatory only if you select one of the following options in the <b>Asset Type</b> field:</p> <ul style="list-style-type: none"> <li>◦ <b>APISUITE</b></li> <li>◦ <b>APITEST</b></li> <li>◦ <b>APISTUB</b></li> <li>◦ <b>POSTMAN</b></li> </ul> <p>The following message is displayed when you select any other asset type than preceding list:</p> <p>You can select Test Environment only if you are running tests of type APISUITE, APITEST, APISTUB, or POSTMAN.</p>
<b>Use Custom Trust Store</b>	<p>Select this checkbox if you used an internal CA certificate and imported the certificate to the custom trust store. You can then enter the file path of your trust store followed by the file name in the <b>Custom Trust Store Path</b> field.</p> <p>Select the <b>Use Custom Password for the Trust Store</b> checkbox if you modified the trust store password, and then enter the new password in the <b>Custom Trust Store Password</b> field.</p> <p> <b>Note:</b> If you did not modify the trust store password, you must clear the <b>Use Custom Password for the Trust Store</b> checkbox.</p>
<b>Variables</b>	<p>Use this field to enter the name of the variable and its value if your test requires variables during the test run time.</p> <p>You must enter the variables in the following format:</p> <p>name_of_the_variable=value_of_the_varibale</p> <p>You can add multiple variables to the test run separated by a semicolon.</p>

Field	Description
	For example, <i>varname1=value1;varname2=value2</i>

- Click **Generate Pipeline Script** and copy the script that you created for the build step to use it in the sample script.

## Results

You have created the pipeline script.

## What to do next

You can configure the **Pipeline** project by using the **Pipeline script** option. See [Configuring the Pipeline project by using the sample script on page 574](#).

## Configuring the Pipeline project by using the sample script

You can configure a **Pipeline** project directly from the Jenkins UI by using the sample pipeline scripts to run test assets from Jenkins.

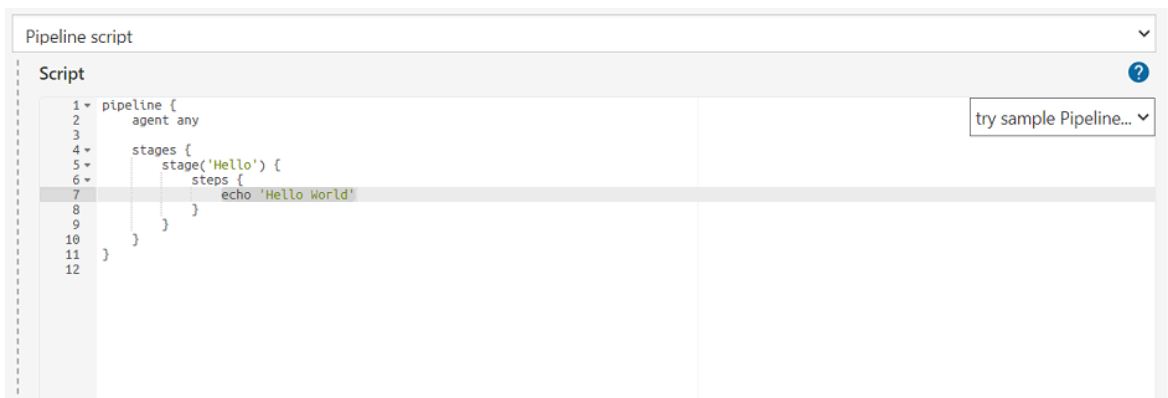
## Before you begin

You must have completed the following tasks:

- Installed the Rational® Test Automation Server Jenkins plugin on the Jenkins server. See [Installing the Rational Test Automation Server Jenkins plug-in on page 563](#).
- Created a pipeline script. See [Creating a pipeline script from Jenkins on page 570](#).

- Open your Jenkins **Pipeline** project from the list.
- Click **Configure**, and then select the **Pipeline** tab.
- Select the **Pipeline script** option in the **Definition** field.
- Select the sample script from the drop-down list.

For example, if you select `Hello World` from the drop-down list, then the sample script is displayed as follows:

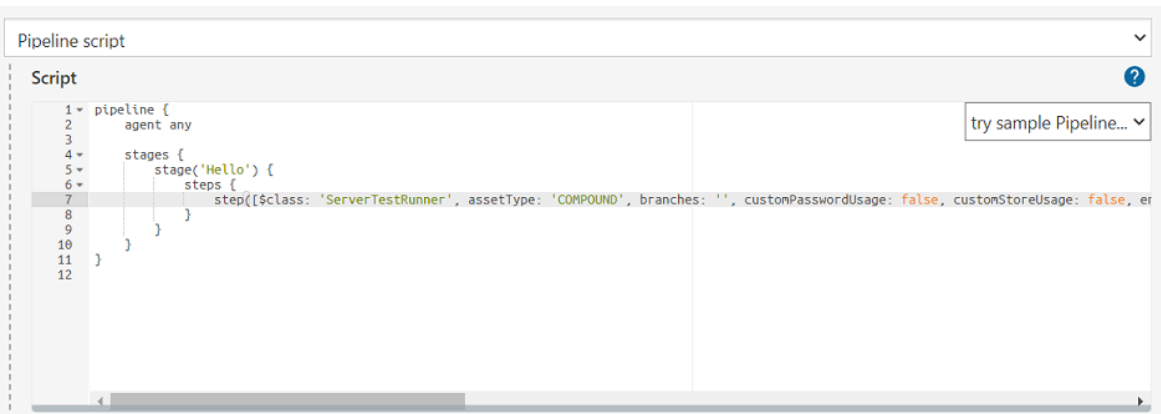


```

Pipeline script
Script
1 pipeline {
2 agent any
3
4 stages {
5 stage('Hello') {
6 steps {
7 echo 'Hello World'
8 }
9 }
10 }
11 }
12
try sample Pipeline...

```

5. Specify the script that you created in place of `echo 'Hello Word'`.



The screenshot shows the Jenkins Pipeline script editor. The script content is as follows:

```

1 pipeline {
2 agent any
3
4 stages {
5 stage('Hello') {
6 steps {
7 step([$class: 'ServerTestRunner', assetType: 'COMPOUND', branches: '', customPasswordUsage: false, customStoreUsage: false, er
8 }
9 }
10 }
11 }
12

```

There is a dropdown menu on the right side of the editor with the text "try sample Pipeline..." and a question mark icon.



**Note:** You can provide a different name for the `stage` in the script.

### Results

You have configured the **Pipeline** project by using the **Pipeline script** option.

### What to do next

You can run test assets from the Jenkins server. See [Running tests from Jenkins on page 576](#).

## Configuring the Pipeline project by using the script from SCM

You can configure the **Pipeline** project to use the pipeline script from the Source Code Management (SCM), and then run test assets that are available in the Rational® Test Automation Server project from the Jenkins **Pipeline** project.

### Before you begin

You must have completed the following tasks:

- Installed the Rational® Test Automation Server Jenkins plugin on the Jenkins server. See [Installing the Rational Test Automation Server Jenkins plug-in on page 563](#).
- Created a pipeline script and committed the script to the Git repository. For more information, refer to the [Jenkins](#) documentation.
- Been granted administrator permission of the Jenkins server to add credentials into Jenkins.
- Added your GitHub credentials into Jenkins. For more information about adding global credentials, refer to the [Jenkins](#) documentation.
- Created a Jenkins **Pipeline** project.

### About this task

If your pipeline scripts are complex, you can then write the script outside of Jenkins UI. You must commit that script file into your Git repository. During the **Pipeline** build process, Jenkins checks out the script file from the Git repository and runs your **Pipeline** project.

1. Open your Jenkins **Pipeline** project from the list.
2. Click **Configure**, and then select the **Pipeline** tab.
3. Select the **Pipeline script from SCM** option in the **Definition** field.
4. Select the **Git** option from the drop-down list in the **SCM** field.
5. Enter the URL of the Git repository in the **Repository URL** field where you stored your pipeline script.
6. Select the credentials of the Git repository from the drop-down list.
7. **Optional:** Enter the branch name of the Git repository in the **Branch Specifier** field.
8. Enter the path of the script that you stored in the Git repository in the **Script path** field.
9. Click **Save**.

### Results

You have configured the **Pipeline** project by using the **Pipeline script from the SCM** option.

### What to do next

You can run test assets from the Jenkins server. See [Running tests from Jenkins on page 576](#).

## Running tests from Jenkins

You can run test assets either from the Jenkins **Freestyle** or the **Pipeline** project on the Jenkins server to test an application under test.

### Before you begin

You must have completed the following tasks:

- Configured the **Freestyle** project, if you want to run test assets from the **Freestyle** project. See [Configuring the Freestyle project on page 566](#).
- Configured the **Pipeline** project either by using the **Pipeline script** or **Pipeline script from SCM**, if you want to run test assets from the **Pipeline script** project. See [Configuring the Pipeline project by using the sample script on page 574](#) or [Configuring the Pipeline project by using the script from SCM on page 575](#).
- Downloaded the latest **Test Results Analyzer** plug-in from the [Jenkinsci](#) repository. For more information about the **Test Results Analyzer** plug-in, refer to the [Jenkins](#) documentation.
- Installed the **Test Results Analyzer** plug-in on the Jenkins server. For more information about installing the plug-in, refer to the [Jenkins](#) documentation.

### About this task

After the test run is complete, you can view the test results in a tabular format when you use the Jenkins **Test Results Analyzer** plug-in. With a TRA-based report, you can filter the results based on the status of the build. The status of the

builds can be `PASSED`, `FAILED`, and `SKIPPED`. You can also identify the failed build from the TRA-based report and take the required action to resolve the build errors. The TRA report also represents the build information in **Line**, **Bar**, and **Pie** graphs so that you can easily read the data.

1. Log in to the Jenkins server.

**Result**

The Jenkins dashboard is displayed.

2. Open your Jenkins **Freestyle** or **Pipeline** project from the list.
3. Click **Build Now** to run the test assets from Jenkins.

**Results**

You have run the test from the Jenkins server.

**What to do next**

You can view the build logs by clicking the build number from the **Build History** pane, and then selecting the **Console Output** option. You can also view the history of test results in a tabular format by clicking the **Test Results Analyzer** option.

You can access the report URLs to view the test run information at any point in time. You can also view the test reports and logs of the test that was run on the Jenkins server from the **Results** page on Rational® Test Automation Server. See [Test results and reports on page 456](#).

## Integration with UrbanCode™ Deploy

You can integrate IBM® Rational® Test Automation Server with IBM® UrbanCode™ Deploy to run test assets as a **Process** from UrbanCode™ Deploy.

When you use UrbanCode™ Deploy for automating the deployments of your applications to an environment during the development cycle, you can create test assets for your application in desktop clients. After you configure the project in Rational® Test Automation Server, you can then run those tests assets from UrbanCode™ Deploy.

You must use the Rational® Test Automation Server UCD plug-in to integrate Rational® Test Automation Server with UrbanCode™ Deploy. Integrating Rational® Test Automation Server with UrbanCode™ Deploy automates the process of test runs. If you have many tests to run at regular intervals, you can use UrbanCode™ Deploy to initiate test runs automatically.

The following table lists the tasks that you must perform to run performance test assets from UrbanCode™ Deploy:

Tasks	Go to...
Create any or all the following types of test assets in desktop clients to test your application: <ul style="list-style-type: none"> <li>• API test assets</li> <li>• Functional test assets</li> <li>• Performance test assets</li> </ul>	<ul style="list-style-type: none"> <li>• <a href="#">Rational® Performance Tester documentation</a></li> <li>• <a href="#">Rational® Functional Tester documentation</a></li> <li>• <a href="#">Rational® Test Workbench documentation</a></li> </ul>
Create a project, and then configure the repository in Rational® Test Automation Server.	<a href="#">Working with projects in a team space on page 616</a>
Install UrbanCode™ Deploy with at least one agent.	<a href="#">IBM UrbanCode™ Deploy documentation.</a>  <a href="#">System Requirements on page 17</a> for information about specific versions of UrbanCode™ Deploy requirements.
Install the Rational® Test Automation Server UCD plug-in on UrbanCode™ Deploy.	<a href="#">Installing the plug-in on UrbanCode Deploy on page 578.</a>
Create a <b>Component</b> in UrbanCode™ Deploy.	<a href="#">Creating a component in UrbanCode Deploy on page 581.</a>
Create a <b>Process</b> for the component in UrbanCode™ Deploy.	<a href="#">Creating a process in UrbanCode Deploy on page 581.</a>
Configure the <b>Process</b> that you created for the component in UrbanCode™ Deploy.	<a href="#">Configuring the process on page 582.</a>
Create a <b>Resource</b> in UrbanCode™ Deploy.	<a href="#">Creating a resource in UrbanCode Deploy on page 587.</a>
Configure the <b>Resource</b> .	<a href="#">Configuring the resource on page 588.</a>
Create an <b>Application</b> in UrbanCode™ Deploy.	<a href="#">Creating an application in UrbanCode Deploy on page 589.</a>
Configure the <b>Application</b> that you want to use to run test assets from UrbanCode™ Deploy.	<a href="#">Configuring the application on page 590.</a>
Run the test assets from UrbanCode™ Deploy.	<a href="#">Running tests from UrbanCode Deploy on page 592.</a>

## Installing the plug-in on UrbanCode™ Deploy

You must install the Rational® Test Automation Server UCD plug-in to run test assets from UrbanCode™ Deploy.

### Before you begin

- You must be familiar with working with UrbanCode™ Deploy.
- You must have performed the following tasks:

- Installed UrbanCode™ Deploy. For more information about the installation, refer to the [IBM Urban Code](#) documentation.
- Been granted access to UrbanCode™ Deploy.
- Downloaded the IBM® Rational® Test Automation Server UCD plug-in from the [IBM WebSphere, Liberty & DevOps Community portal](#).

For more information about specific versions of plug-in, see [Integration plug-in compatibility matrix on page 543](#).

1. Log in to UrbanCode™ Deploy, if you are not already logged in.

**Result**

The UrbanCode™ Deploy dashboard is displayed.

2. Click **Settings**.
3. Click **Automation Plugins** from the **Automation** pane.
4. Click **Load plug-in**.
5. Click **Choose File** to locate and open the compressed Rational® Test Automation Server UCD plug-in file.



**Remember:** You must not extract the Rational® Test Automation Server UCD plug-in compressed file contents.

6. Click **Submit**.

**Result**

The installed plug-in is displayed on the **Automation Plugins** tab.

**Results**

You have installed the Rational® Test Automation Server UCD plug-in on UrbanCode™ Deploy.

**What to do next**

You must create a **Component** in UrbanCode™ Deploy. See [Creating a component in UrbanCode Deploy on page 581](#).

## Using a custom trust store

You can use a custom trust store in the IBM® Rational® Test Automation Server UCD plug-in file to establish a trusted and secure connection between the UCD server and Rational® Test Automation Server.

**Before you begin**

You must have completed the following tasks:

- Saved the Certificate Authority (CA) assigned to Rational® Test Automation Server.



**Note:** You can contact the administrator of Rational® Test Automation Server if you do not have a copy of the CA.

- Installed Java on the computer from where you access the Rational® Test Automation Server UI.

### About this task

If the SSL certificate assigned to Rational® Test Automation Server is signed by an internal Certified Authority (CA), then you must download and import the CA certificate to a custom trust store. You can then use the custom trust store in the **IBM® Rational® Test Automation Server test** process step to establish a trusted and secure connection between the UCD server and Rational® Test Automation Server.



**Note:** If the internal CA certificate is already imported to the default trust store that is used by the UCD server, you need not use a custom trust store.

1. Locate the default trust store file (`cacerts` file) from the JRE directory.  
For example: `C:\Program Files\Java\jdk-18.0.1.1\lib\security`
2. Copy the `cacerts` file to a location of your choice on your computer.
3. Place the CA assigned to Rational® Test Automation Server in the same location where you saved the `cacerts` file.
4. Run the following command from the command-line interface or terminal to import the CA of Rational® Test Automation Server to custom trust store:

```
keytool -import -trustcacerts -file {path to the downloaded CA with the file extension} -alias {label for the certificate} -keystore {path to the trust store cacerts file}
```

For example,

```
keytool -import -trustcacerts -file C:\Users\Desktop\cert\ca-master-ibm.crt -alias "rtasca" -keystore C:\Users\Desktop\cert\cacerts
```



**Note:** The default password of the trust store is *changeit*. It remains the same for the custom trust store. If you want to change the password, you can run the following command, and then enter the new password:

```
keytool -storepasswd -keystore {path to the trust store cacerts file}
```

For example, `keytool -storepasswd -keystore C:\Users\Desktop\cert\cacerts`

### Results

You have successfully imported the downloaded CA to the custom trust store.

### What to do next



You can then run the tests that are available in your Rational® Test Automation Server project from the UCD server.

## Creating a component in UrbanCode™ Deploy

You must create a component to include artifacts and processes. The artifacts include runnable files, images, databases, configuration instructions. Whereas the processes define the activities that components can perform.

### Before you begin

- You must be familiar with working with UrbanCode™ Deploy.
- You must have been granted access to UrbanCode™ Deploy.

1. Log in to UrbanCode™ Deploy, if you are not already logged in.

#### Result

The UrbanCode™ Deploy dashboard is displayed.

2. Click **Components**, and then click **Create Component**.
3. Enter a name for the component in the **Name** field.
4. Enter the details in the other optional fields based on your requirement, and then click **Save**.

#### Result

The component that you created is displayed.

### Results

You have created the component in UrbanCode™ Deploy.

### What to do next

You must create a process for the component in UrbanCode™ Deploy. See [Creating a process in UrbanCode Deploy on page 581](#).

---

Related information

[IBM UrbanCode Deploy Documentation](#)

## Creating a process in UrbanCode™ Deploy

You must create a process for the component to include step properties for the test that you want to run from UrbanCode™ Deploy.

### Before you begin

- You must be familiar with working with UrbanCode™ Deploy.
- You must have performed the following tasks:

- Been granted access to UrbanCode™ Deploy.
- Created a component in UrbanCode™ Deploy. See [Creating a component in UrbanCode Deploy on page 581](#).

1. Log in to UrbanCode™ Deploy, if you are not already logged in.

**Result**

The UrbanCode™ Deploy dashboard is displayed.

2. Click **Components**.

**Result**

A list of components that are available in UrbanCode™ Deploy is displayed.

3. Select the component from the list for which you want to create a process.
4. Click the **Processes** tab, and then click **Create Process**.

**Result**

The **Create Process** dialog is displayed.

5. Enter a name for the process in the **Name** field.
6. Select **Operational (No Version Needed)** from **Process Type** drop-down list.
7. Verify the **Default Working Directory** field.

The **Default Working Directory** field defines the location that the agent uses to run the process. The default value is `${p:resource/work.dir}/${p:component.name}`.

Where `${p:resource/work.dir}` is the default working directory for the agent and `${p:component.name}` is the name of the component.

8. Click **Save**.

**Result**

The process that you created is listed in the **Processes** tab and the **Design** tab for the process is displayed.

## Results

You have created the process for the component in UrbanCode™ Deploy.

## What to do next

You must configure the process in UrbanCode™ Deploy. See [Configuring the process on page 582](#).

---

Related information

[IBM UrbanCode Deploy Documentation](#)

## Configuring the process

You must configure the process that you created for the component to organize the steps in the process, specify the properties of the steps, and connect them.

### Before you begin

- You must be familiar with working with UrbanCode™ Deploy.
- You must have performed the following tasks:
  - Been granted access to UrbanCode™ Deploy.
  - Created a component in UrbanCode™ Deploy. See [Creating a component in UrbanCode Deploy on page 581](#).
  - Created a process for the component in UrbanCode™ Deploy. See [Creating a process in UrbanCode Deploy on page 581](#).
  - Generated an offline user token from Rational® Test Automation Server. See [Generating an offline token on page 600](#).
  - Added the tests assets to the project on Rational® Test Automation Server.

### About this task

When you open any process to configure, the process is displayed in the process editor. The process editor lists the plug-ins and steps. The required **Start** and **Finish** steps represent the beginning and the end of the process and steps are automatically placed on the design area.

1. Log in to UrbanCode™ Deploy, if you are not already logged in.

#### Result

The UrbanCode™ Deploy dashboard is displayed.

2. Click **Components**.

#### Result

A list of components that are available in UrbanCode™ Deploy is displayed.

3. Select the component from the list in which you created the process.

4. Click the **Processes** tab.

#### Result

A list of processes that are available for the component is displayed.

5. Select the process from the list that you want to configure.

#### Result

The **Design** tab for the process is displayed.

6. Click **IBM Rational Test Workbench**, and then **IBM Rational Test Automation Server** from the left menu.

7. Drag the **Run IBM® Rational® Test Automation Server test** step, and then drop it into the design area.



**Note:** The selected test must be placed between **Start** and **Finish** steps.

8. Specify the properties for the selected test by performing the following steps:


- a. Click the **Edit** icon.

#### Result

The **Edit Properties for Run IBM Rational Test Automation Server test** dialog is displayed.



- b. Specify the properties for the selected test step by referring to the following tables:


The following table lists the required fields that you must provide to run the test from UrbanCode™ Deploy:

Fields	Action
<b>Name</b>	Enter the name for the test step.
<b>IBM Rational Test Automation Server URL</b>	Enter the URL of Rational® Test Automation Server.
<b>Offline Token</b>	Enter the offline token that you generated from Rational® Test Automation Server.
<b>Team Space Name</b>	Enter the name of the team space that contains the project.   <b>Note:</b> The license for the team space must be configured and you must be a member of that team space.
<b>Project Name</b>	Enter the name of the project that are available in the team space.
<b>Repository Link</b>	Enter the URL of the Git repository that you added to your project in Rational® Test Automation Server.
<b>File Path</b>	Enter the path of the test assets that you want to run.  You can find the path of the test assets from the <b>Execution</b> page in Rational® Test Automation Server

The following table lists the optional fields that you can provide to run the test from UrbanCode™ Deploy:

Fields	Description
<b>Branch Name</b>	Use this field to enter the name of the branch in the Git repository where you stored test assets.

Fields	Description
<b>Custom Trust Store Password</b>	<p>Use this field to enter the trust store password if you have modified the password while creating the custom trust store.</p> <p> <b>Note:</b> If you have not modified the trust store password, you can retain the <b>Custom Trust Store Password</b> field blank.</p>
<b>Custom Trust Store Path</b>	<p>Use this field to enter the file path of your trust store followed by the file name if Rational® Test Automation Server uses an internal CA certificate and you have imported the certificate to a custom trust store.</p>
<b>Datasets</b>	<p>Use this field to enter the path to the dataset if you want to replace the values of the dataset during a test run.</p> <p>You must ensure that both original and new datasets are in the same workspace and have the same column names. When you enter a value for the <b>Datasets</b> field, you must also include the path to the dataset. The path must be in the following format:</p> <pre data-bbox="480 1010 1412 1077">/project_name/ds_path/original_ds.csv:/project_name/ds_path/new_ds.csv</pre> <p> <b>Note:</b> You can override multiple datasets that are saved in a different project by adding multiple paths to the dataset separated by a semicolon.</p>
<b>Environment</b>	<p>Use this field to enter the environment details for the following types of test assets:</p> <ul style="list-style-type: none"> <li>▪ APISUITE</li> <li>▪ APITEST</li> <li>▪ APISTUB</li> </ul>
<b>Labels</b>	<p>Use this field to add labels to test results when the test run is complete.</p> <p>You can add multiple labels to a test result separated by a comma.</p> <p>For example, <i>label1, label2</i></p> <p>After the test run is complete, then the values provided in the <b>Labels</b> field are displayed on the <b>Results</b> page of Rational® Test Automation Server.</p>

Fields	Description
<b>Secrets Collection Name</b>	<p>Use this field to enter the name of the Secret if you created secrets collections for your project and to use Secrets at test run time.</p> <p>This field is mandatory only if you want to run the following types of test assets:</p> <ul style="list-style-type: none"> <li>▪ APISUITE</li> <li>▪ APITEST</li> <li>▪ APISTUB</li> </ul>
<b>Show Hidden Properties</b>	<p>Select this checkbox if you want to use an HTTP Proxy to connect to Rational® Test Automation Server.</p> <p>After you select the <b>Show Hidden Properties</b> checkbox, you can configure the HTTP Proxy by using the following fields:</p> <ul style="list-style-type: none"> <li>▪ <b>HTTP Proxy Host:</b> Use this field to enter the hostname of the HTTP proxy that you want to connect to Rational® Test Automation Server.</li> <li>▪ <b>HTTP Proxy Port:</b> Use this field to enter the port number where the HTTP proxy is used to listen to both HTTP and HTTPS traffic.</li> <li>▪ <b>HTTP Proxy User name:</b> Use this field to provide the username for the HTTP proxy.</li> <li>▪ <b>HTTP Proxy Password:</b> Use this field to enter a valid password for the user that you specified in the <b>HTTP Proxy User name</b> field.</li> </ul> <p> <b>Note:</b> You can use the <b>Show Hidden Properties</b> field only when you want to run the following types of test assets:</p> <ul style="list-style-type: none"> <li>▪ APISUITE</li> <li>▪ APITEST</li> <li>▪ APISTUB</li> </ul>
<b>Variables</b>	<p>Use this field to enter the name of the variable and its value if your test requires variables during the test run time.</p> <p>You must enter the variables in the following format:</p> <pre>name_of_the_variable=value_of_the_varibale</pre> <p>You can add multiple variables to the test run separated by a semicolon.</p> <p>For example, <i>varname1=value1;varname2=value2</i></p>



**Note:** In addition to optional fields, you can also use the following fields from UrbanCode™

Deploy to configure your test run:

- **Working Directory**
- **Precondition**
- **Post Processing Script**
- **Use Impersonation**
- **Auth Token Restriction**

You can accept the default values or change the values based on your requirements. For more information about these fields, see the related links.

c. Click **OK** to save the properties for the test.

9. Click **Save** in the design area.

## Results

You have configured the process for the component in UrbanCode™ Deploy.

## What to do next

You must create a resource in UrbanCode™ Deploy. See [Creating a resource in UrbanCode Deploy on page 587](#).

---

Related information

[IBM UrbanCode Deploy Documentation](#)

[Process step preconditions](#)

[Post-processing scripts](#)

[User impersonation for process steps](#)

[Restricting authentication tokens](#)

## Creating a resource in UrbanCode™ Deploy

You must create a resource to associate agents with components that you created in UrbanCode™ Deploy.

### Before you begin

- You must be familiar with working with UrbanCode™ Deploy.
  - You must have been granted access to UrbanCode™ Deploy.
1. Log in to UrbanCode™ Deploy, if you are not already logged in.

### Result

The UrbanCode™ Deploy dashboard is displayed.

2. Click **Resources**, and then click **Create Top-Level Group**.

### Result

The **Create Resource** dialog is displayed.

3. Enter a name for the resource in the **Name** field.
4. Click **Save**.

### Results

You have created the resource in UrbanCode™ Deploy.

### What to do next

You must configure the resource. See [Configuring the resource on page 588](#).

---

Related information

[IBM UrbanCode Deploy Documentation](#)

## Configuring the resource

You must configure the resource to add an agent and associate the agent with the component.

### Before you begin

- You must be familiar with working with UrbanCode™ Deploy.
- You must have performed the following tasks:
  - Been granted access to UrbanCode™ Deploy.
  - Created a component in UrbanCode™ Deploy. See [Creating a component in UrbanCode Deploy on page 581](#).
  - Created a resource in UrbanCode™ Deploy. See [Creating a resource in UrbanCode Deploy on page 587](#).

1. Log in to UrbanCode™ Deploy, if you are not already logged in.

### Result

The UrbanCode™ Deploy dashboard is displayed.

2. Click **Resources**.

### Result

A list of resources that are available in UrbanCode™ Deploy is displayed.

3. Perform the following steps to add an agent to the resource:
  - a. Click the resource from the list for which you want to add an agent.
  - b. Click the **Actions** icon from the last column, and then click **Add Agent**.
  - c. Select the agent from the drop-down list.





**Note:** The **Name** field is auto populated with the name of the agent.

d. Click **Save**.

**Result**

The selected agent is added to the resource and you can view the status of the agent in the **Status** column.

4. Perform the following steps to add a component to the agent:

a. Click the agent from the list for which you want to add a component.

b. Click the **Actions** icon from the last column, and then click **Add Component**.

c. Select the component from the drop-down list.



**Note:** The **Name** field is auto populated with the name of the component.

d. Click **Save**.

**Result**

The selected component is added to the agent.

## Results

You have configured the resource in UrbanCode™ Deploy.

## What to do next

You must create an application. See [Creating an application in UrbanCode Deploy on page 589](#).

---

Related information

[IBM UrbanCode Deploy Documentation](#)

## Creating an application in UrbanCode™ Deploy

You must create an application to fetch all the components together that you want to deploy.

### Before you begin

- You must be familiar with working with UrbanCode™ Deploy.
- You must have been granted access to UrbanCode™ Deploy.

1. Log in to UrbanCode™ Deploy, if you are not already logged in.

**Result**

The UrbanCode™ Deploy dashboard is displayed.

2. Click **Applications**.

3. Click **Create Applications**, and then **New Applications**.
4. Enter a name for the application in the **Name** field.

**Result**

The **Environments** page for the application that you created is displayed.

**Results**

You have created the application in UrbanCode™ Deploy.

**What to do next**

You must configure the application. See [Configuring the application on page 590](#).

---

Related information

[IBM UrbanCode Deploy Documentation](#)

## Configuring the application

You must configure the application to associate resources with environments and define processes to run test assets.

**Before you begin**

- You must be familiar with working with UrbanCode™ Deploy.
- You must have performed the following tasks:
  - Been granted access to UrbanCode™ Deploy.
  - Created a component in UrbanCode™ Deploy. See [Creating a component in UrbanCode Deploy on page 581](#).
  - Created a process for the component in UrbanCode™ Deploy. See [Creating a process in UrbanCode Deploy on page 581](#).
  - Configure the process for the component in UrbanCode™ Deploy. See [Configuring the process on page 582](#).
  - Created a resource in UrbanCode™ Deploy. See [Creating a resource in UrbanCode Deploy on page 587](#).
  - Configured the resource in UrbanCode™ Deploy. See [Configuring the resource on page 588](#).
  - Created an application in UrbanCode™ Deploy. See [Creating an application in UrbanCode Deploy on page 589](#).

1. Log in to UrbanCode™ Deploy, if you are not already logged in.

**Result**

The UrbanCode™ Deploy dashboard is displayed.

2. Click **Applications**.

**Result**

A list of applications that are available in UrbanCode™ Deploy is displayed.

3. Click the application that you want to configure from the **Name** column.

**Result**

The **Environments** page for the selected application is displayed.

4. Perform the following steps to create an environment for the application that you selected:
  - a. Click **Create Environment**.
  - b. Enter a name for the environment in the **Name** field.
  - c. Click **Save**.

5. Perform the following steps to configure resources to the environment:
  - a. Click the environment that you created.
  - b. Click **Add Base Resources**.

**Result**

A list of resources that are available in UrbanCode™ Deploy is displayed.

- c. Select the checkbox to add resources to the environment.
- d. Click **Save**.

**Result**

You can view the corresponding agent and the component that you added for the resource by using the **Expand** icon.

6. Perform the following steps to add the component to the application:
  - a. Click **Applications**, and then select your application from the list.
  - b. Click the **Components** tab, and then **Add Components**.
  - c. Select the checkbox from the drop-down list to add components to the application.
  - d. Click **Save**.

7. Perform the following steps to create a process for the application:
  - a. Click the **Processes** tab, and then **Create Process**.
  - b. Enter a name for the process in the **Name** field.
  - c. Click **Save**.

**Result**

The **Design** tab for the process that you created is displayed.

8. Drag the component process listed under the **Component Process Steps** option from the left navigation pane and drop it into the design area.
9. Select the component process from the drop-down list in the **Operational (No Version Needed) Process** field.
10. Click **Save**.

11. Click the **Edit** icon, and then change the name of the properties.
12. Click **OK**, and then click **Save**.

## Results

You have configured the application to run test assets from UrbanCode™ Deploy.

## What to do next

You can run test assets from UrbanCode™ Deploy. See [Running tests from UrbanCode Deploy on page 592](#).

---

Related information

[IBM UrbanCode Deploy Documentation](#)

## Running tests from UrbanCode™ Deploy

You can run test assets from UrbanCode™ Deploy as a **Process** to test your application under test.

### Before you begin

- You must be familiar with working with UrbanCode™ Deploy.
- You must have performed the following tasks:
  - Been granted access to UrbanCode™ Deploy.
  - Created a component in UrbanCode™ Deploy. See [Creating a component in UrbanCode Deploy on page 581](#).
  - Created a process for the component in UrbanCode™ Deploy. See [Creating a process in UrbanCode Deploy on page 581](#).
  - Configure the process for the component in UrbanCode™ Deploy. See [Configuring the process on page 582](#).
  - Created a resource in UrbanCode™ Deploy. See [Creating a resource in UrbanCode Deploy on page 587](#).
  - Configured the resource in UrbanCode™ Deploy. See [Configuring the resource on page 588](#).
  - Created an application in UrbanCode™ Deploy. See [Creating an application in UrbanCode Deploy on page 589](#).
  - Configured the application in UrbanCode™ Deploy. See [Configuring the application on page 590](#).
  - Started the UrbanCode™ Deploy agent. For more information, refer to the [IBM UrbanCode Deployment documentation](#).

1. Log in to UrbanCode™ Deploy, if you are not already logged in.

**Result**

The UrbanCode™ Deploy dashboard is displayed.

2. Click **Applications**.

**Result**

A list of applications that are available in UrbanCode™ Deploy is displayed.

3. Click the application that you configured for the test run.

4. Click the **Request Process** icon.

**Result**

The **Run Application Process** page is displayed.

5. Select the process that you created for the application which contains the test assets from the **Process** drop-down list.

6. Click **Submit**.

**Result**

The UrbanCode™ Deploy dashboard shows the progress of the process request.

## Results

You have run the performance test assets as **Process** from UrbanCode™ Deploy.

## What to do next

After the process completes, you can view the status of the process displayed as follows:

- **Success:** When the test run is successful
- **Failed:** When the test run is failed

You can view the details of the test run in the **Execution Log** section. The **Reports information** section on the `Log` file displays the names of the report along with its corresponding URLs.

## Integration with Xray test management for Jira

When you use Jira as a change management system on Rational® Test Automation Server and you use Xray as the test management application in Jira, you can integrate Xray with Rational® Test Automation Server. With this integration, you can export the test results of the tests that run on Rational® Test Automation Server to your Jira project. You can then view and use the test results in your project in Jira.

## Prerequisites

Before you integrate Xray with Rational® Test Automation Server, you must have completed the following tasks:

- Configured Jira as a change management system on Rational® Test Automation Server. See [Configuration of Atlassian Jira as a change management system on page 533](#).
- Installed the Xray test management app in Jira.



**Note:** The Xray integration is supported only when the Xray test management application is installed on a Jira cloud.

- Ensured that you obtained the access and authentication details to Jira from the Jira administrator.
- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created a project in your team space in Rational® Test Automation Server. See [Adding a project on page 619](#).

## Task flow

You can follow the tasks that are listed in the following table when you want to export test results to your Xray test project in Jira. The table also provides you the links to the information about the tasks.

Task	Go to...
Generate the Xray API keys for your username on Jira.	<a href="#">Generating API keys for the Xray app in Jira on page 594</a>
Use the generated Xray API keys to configure the Xray integration in Rational® Test Automation Server.	<a href="#">Configuring the Xray integration on page 595</a>
Select an issue in your Xray project in Jira as a location to export the test results of a test run. You must select the issue while you configure a test run on Rational® Test Automation Server.	<a href="#">Selecting a Jira Xray issue to export test results on page 597</a>

You can manage the Xray settings that you configured in Rational® Test Automation Server. You can edit the saved configuration or you can delete the configuration. For more information, see [Managing the Xray configuration on page 598](#).

## Generating API keys for the Xray app in Jira

You must generate API keys for the Xray app in Jira and use them to configure the Xray integration in Rational® Test Automation Server. The Xray API keys are used by Jira to authenticate the requests from Rational® Test Automation Server.

### Before you begin

You must have completed the following tasks:

- Read about the prerequisite tasks in [Prerequisites on page 593](#).
- Verified that you can access Jira.

- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Created a project in your team space in Rational® Test Automation Server. See [Adding a project on page 619](#).

1. Log in to Jira.
2. Click **Settings > Apps**.

The apps that are installed in Jira are listed in the navigation pane.

3. Find the **Xray App** in the navigation pane, and then click the **API Keys** option.



The **API Keys** page is displayed.

4. Click **Create API Key**.

The **Create API Key** dialog is displayed.

5. Enter your username in the **User** field, and then select the username that is displayed.
6. Click **Generate**.

The API keys are generated and displayed under the **Client Id** and **Client Secret** fields.

7. Click the **Copy** icon  in the **Client Id** field to copy the key, and then paste the copied key into a text file.
8. Click the **Copy** icon  in the **Client Secret** field to copy the key, and then paste the copied key into a text file.

## Results

You have generated the Xray API keys in Jira.

## What to do next

You must use the Xray API keys to configure the Xray integration in Rational® Test Automation Server. See [Configuring the Xray integration on page 595](#).

## Configuring the Xray integration

You can set up the Xray integration when you want to export test results from Rational® Test Automation Server to an issue in your Xray project in Jira. You must configure the Xray integration in your project in Rational® Test Automation Server before you select the Xray project in Jira as the location to export the test results after the test runs in Rational® Test Automation Server.

## Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Generated the Xray API keys in Jira and saved the keys in a text file. See [Generating API keys for the Xray app in Jira on page 594](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Click **Active projects > My projects > *project\_name*** to open the project that contains the test assets.

The **Overview** page of the project is displayed.

3. Click **Manage > Integration** in the navigation pane.

The **Integration** page is displayed.

4. Click the **TEST MANAGEMENT** tab, if it is not already open.

5. Click **Add Integration > Jira/Xray Software**.

The **Add Xray Integration** dialog is displayed.

6. Perform the following steps:

- a. Enter the Xray API key that was generated for the **Client Id** and copied into the text file, in the **Client Id** field.
- b. Enter the Xray API key that was generated for the **Client Secret** and copied into the text file, in the **Client Secret** field.
- c. Click **Save**.

The Xray configuration is saved and displayed on the **TEST MANAGEMENT** page.

## Results

You configured the Xray integration on Rational® Test Automation Server.

## What to do next

You can perform the following tasks:

- View the following information on the **TEST MANAGEMENT** tab that you used to configure the Xray integration:
  - The URL of the Jira server on which the Xray test management app is installed.
  - The project that you selected in Jira.



- The Xray API key that is used as the Client ID.
  - The status of the Xray configuration.
- Select an issue in Jira as the location to which you can export the test results of a test run while you configure a run of the test on Rational® Test Automation Server. See [Selecting a Jira Xray issue to export test results on page 597](#).

## Selecting a Jira Xray issue to export test results

After you configure the Xray integration on Rational® Test Automation Server, you can select a Jira Xray issue in Jira or create a new issue in Jira to export the test results. The test results are exported to the selected issue after the tests run on Rational® Test Automation Server. You can then view and track the test results in your Jira project.

### Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Configured the Xray integration by using the Xray API keys that are generated for your username. See [Configuring the Xray integration on page 595](#).
- Ensured that the **Execute test asset** dialog is displayed and you have configured the other settings for a test run in Rational® Test Automation Server.

### About this task

When you configure a test run in Rational® Test Automation Server, you can select a Jira issue as the location where you want the test results to be exported. After the test is run, the results are exported to the Jira issue. You can then view and track the test results in the Jira issue from your project in Jira.

1. Click the **RESULTS** tab in the **Execute test asset** dialog.


The **RESULTS** tab is displayed.



**Note:** The **Export report options** field is enabled only if you configured the Xray integration with the Xray API keys that are generated for your username in Jira.

2. Perform the actions listed in the following table to select an option to export the test results:

Export options	Results in...	Actions
Jira Xray	The test results of the test that you configure for a run are exported to the selected issue after the test	Perform the following steps: <ol style="list-style-type: none"> <li>a. Select the <b>Jira Xray</b> option.</li> </ol> <p>The <b>Select issue</b> field is displayed.</p>

Export options	Results in...	Actions
	run is completed. The results are exported to the selected issue that exists in your Xray project in Jira.	<ul style="list-style-type: none"> <li>b. Click the field to view the list of issues in the Xray project in Jira. Alternatively, you can enter the issue key in the Jira Xray project.</li> <li>c. Select the issue to which you want to export the test results.</li> <li>d. Continue with the next step.</li> </ul>
New Jira item	The test results of the test that you configure for a run are exported to a new Jira issue. The issue is created in your project in Jira after the test run is completed.	Perform the following steps: <ul style="list-style-type: none"> <li>a. Select the <b>New Jira item</b> option.</li> <li>b. Continue with the next step.</li> </ul>
None	The test results are not exported.  <b>Note:</b> This is the default option.	Perform the following steps when: <ul style="list-style-type: none"> <li>◦ You do not want to export the test results.</li> <li>◦ You selected a Jira issue in a previous run of the same test, and you do not want to export results of the current run.</li> </ul> <ul style="list-style-type: none"> <li>a. Select the <b>None</b> option.</li> <li>b. Continue with the next step.</li> </ul>

### Results

You selected an issue in your Xray project or created an issue in Jira as a location to export the test results of a test run on Rational® Test Automation Server.

### What to do next

After the test run is completed, you can go to the **Results** page and expand the test result to view the Xray issue key to which the test results were exported. You can then click the Xray issue key link to open the issue and view the details of the issue in Jira.

## Managing the Xray configuration

You can manage the configuration of the Xray integration that you set in Rational® Test Automation Server when you want to modify the API keys configured or when you want to delete the existing Xray configuration.

### Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Project Owner* or *Tester* in the project. See [Managing access to server projects on page 627](#).
- Ensured that you obtained the access and authentication details to Jira from the Jira administrator.
- Configured the Xray API keys for the Xray integration on Rational® Test Automation Server. See [Configuring the Xray integration on page 595](#).
- Regenerated the Xray API keys in Jira by following the steps to generate the Xray API keys in [Generating API keys for the Xray app in Jira on page 594](#).

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Click **Active projects > My projects > project\_name** to open the project that contains the test assets.


The **Overview** page of the project is displayed.

3. Click **Manage > Integration** in the navigation pane.

The **Integration** page is displayed.

4. Click the **TEST MANAGEMENT** tab, if it is not already open.

The details about the Xray integration that is configured are displayed.

5. Click the **Open action menu** icon  of the Xray integration that you want to manage.

The **Update credentials** and **Delete configuration** options are displayed in the list.

6. Perform any of the following actions when you want to modify or delete the Xray configuration:
  - To modify the Xray API keys that were set previously in the Xray configuration, go to [Step 7 on page 599](#).
  - To delete the Xray configuration, go to [Step 8 on page 599](#).

7. Perform the following steps to modify the Xray API keys:

- a. Select **Update credentials** as the option from the list.

The **Update Xray Integration** dialog is displayed.

- b. Enter the Xray API key that was generated for the **Client Id** and copied into the text file, in the **Client Id** field.

- c. Enter the Xray API key that was generated for the **Client Secret** and copied into the text file, in the **Client Secret** field.

- d. Click **Update** to save the updated Xray API keys in the Xray configuration.

The Xray configuration is updated and a confirmation message is displayed.

8. Perform the following steps to delete the existing Xray configuration:

- a. Select **Delete** as the option from the list.

The **Delete Jira server configuration** dialog is displayed.

b. Click **Delete**.

The Xray configuration is deleted and a confirmation message is displayed.



**Note:** The links to the issues that you created to export the test results are not deleted from the results panel in Rational® Test Automation Server even after you delete the Xray configuration.

## Results

You achieved the following results:

- Updated the Xray configuration.
- Deleted the Xray configuration.

# Managing access to IBM® Rational® Test Automation Server

Desktop clients and third-party integrations use offline user tokens to connect to Rational® Test Automation Server.

You can configure the desktop clients and third-party integrations to access Rational® Test Automation Server by using the offline user tokens created from the server.

Offline user tokens are used in the following cases:

- To retrieve secrets configured on the server to be used in certain tests in Rational® Integration Tester.
- To enable publishing of test results and reports from desktop clients to Rational® Test Automation Server.
- To enable the Resource Monitoring Service in Rational® Performance Tester to monitor a data source during a run on Rational® Test Automation Server.
- To enable agents to connect with Rational® Test Automation Server.
- To enable HTTP proxies to connect with Rational® Test Automation Server.
- To enable third-party integrations.


## About this task

From Rational® Test Automation Server, you can create an offline user token, copy it, and then use that token to enable connections to be authenticated by Rational® Test Automation Server.

You can also delete all the tokens, when you no longer need access or you want to prevent access to the server.

Creating a token

- Follow these steps:

1. Click the **User** icon  from the menu bar, and then select **Create Token**.

The Create Offline User Token dialog is displayed.



**Note:** This token is not accessible after you copy it as the dialog box closes.

2. Copy the token and paste it into a private location for ongoing reference.

The dialog closes.

3. Paste the token into the desktop client UI or in the application where this token is required.


See the following table for a list of where you can use the tokens and where you can find more information.

**Table 16. Using the offline user tokens to access Rational® Test Automation Server**

Use the offline user token in the following places	More information
Rational® Integration Tester	See <a href="#">Rational® Integration Tester and Rational® Test Automation Server..</a>
Rational® Performance Tester	See <a href="#">Publishing test results to the server..</a> See <a href="#">Enablement of Resource Monitoring services for a schedule..</a>
Rational® Functional Tester	See <a href="#">Publishing test results to the server..</a>
Third-party integrations	See <a href="#">Integrations with other applications on page 543.</a>

### Deleting tokens

- Follow these steps:

1. Click the **User** icon  from the menu bar and select **Delete Tokens**.

The Delete All Offline User Tokens dialog is displayed.

2. Inform the desktop client users that the tokens were deleted. Fix any test automation scripts that used tokens.

## Working with IBM® Rational® Test Automation Server REST services

When you want to use the IBM® Rational® Test Automation Server REST APIs in your test automation scripts, you can view the REST APIs that are exposed by IBM® Rational® Test Automation Server. You can also run the APIs from within your project in IBM® Rational® Test Automation Server.

All REST services are accessible from the base URL of IBM® Rational® Test Automation Server and are defined by using Swagger. A navigable user interface lists the available set of APIs at `<server_URL>/#/initial/Admin/docs`. This Swagger file describes the individual APIs along with details of the parameters and response bodies.

You must be assigned the role of an administrator to be able to view or run the REST APIs.

After you log into IBM® Rational® Test Automation Server with your credentials that has administrator privileges, you must go to the following URL: `<server_URL>/#/initial/Admin/docs`

The Swagger docs that are available can be viewed from the drop down list.

You can select the Swagger doc for the REST APIs that you want to view or use. For example, you can select the **/jobservicev2/pi-docs** to view the APIs for the execution micro-service.

You can perform the following steps to view or run the API:

1. Expand the section to view the APIs that are available.
2. Expand each of the APIs to view the details such as the description, examples, parameters, and responses.
3. Run the API by clicking the **Try it out** option and then enter the required values for the parameters.
4. Click **Execute** to start the API.

You can view the response after the API runs.

You can view the API doc for the execution micro-service that is extracted in this [PDF](#) document.

# Chapter 7. Test Manager Guide

This guide describes how to manage and track your overall test effort. This guide is intended for test managers.

---

Related information

[Managing access to server projects on page 627](#)

## Managing team spaces

You might want to become a team space member, add members to your team space, change a user role in a team space, modify the team space, or delete a team space.

Each team space is associated with at least one *Team Space Owner*. You can assign multiple owners to your team space. Any user who becomes a member of a team space and is assigned the role of a *Member*, *Project Creator*, *Architect*, or *Team Space Owner* of a team space, can access the team space.

As an owner of a team space, after you create a team space, you must configure the team space license. You can then add members to the team space and can assign different roles to each member of the team space. You can update or delete a team space. You can also add a repository to a team space to store a system model and edit the repository settings. Apart from adding a repository, you can add, update, or delete the Resource Monitoring agent if no longer required.

As an *Architect* of a team space, you can manage a team space repository and design a system model.

As a *Project Creator* of a team space, you can create a project in the team space.

As a licensed user who is assigned with a *Member*, *Project Creator*, *Architect*, or *Team Space Owner* role of a team space, you can add a Docker host to your team space. You can view the configured Docker, agents, or intercepts of your team space. You can edit only those Docker hosts that you created.

## Viewing team spaces

After you log in to IBM® Rational® Test Automation Server, you can view all the existing team spaces on the **Team Space Dashboard** page.

### About this task

As a member of a team space, you can view your team spaces under **My Team Spaces**. You can also view the team spaces that are created by other users as the public team space and the Permissive team spaces under **Other Team Spaces**.

If the initial team space does not exist, then you can view the **Team Space Dashboard** without any navigation pane.

You as a member of a team space, when you want to use that team space immediately after you log in to Rational® Test Automation Server, you can suffix the alias in the application URL.

For example, if the alias of the team space is `first`, then you must use the following Rational® Test Automation Server URL in the address bar:

```
<server url>/#/first
```

1. Log in to Rational® Test Automation Server.

**Result**

The **Projects** page of the initial team space is displayed.

2. Click the **Settings** icon  from the left navigation pane, and then click **Join another team space**.

**Result**

The **Team Space Dashboard** is displayed.

You can view the team spaces in the following panels:

- **My Team Spaces:** You can view all the team spaces that you created. You can also view the team spaces in which you are a member.
- **Other Team Spaces:** You can view all the team spaces that are created by others.



**Note:** You cannot view the Restrictive private type team spaces.

## Results

You have viewed the team spaces that are created by you and other users on the **Team Space Dashboard**.

## What to do next

You can select a team space and can send a request to become a member of that team space. See [Becoming a team space member on page 611](#).

## Adding members to a team space

When you create a team space to manage projects, you must add users to your team space so that they can create and access the projects.

### Before you begin

- As an administrator, you are assigned the default role of a *Team Space Owner*. If you log in to Rational® Test Automation Server as another user, then you must have been assigned the role of a *Team Space Owner*.
- As an owner of a team space, you must have completed the following tasks:
  - Created a team space. See [Creating a team space on page 129](#).
  - Configured a license for the team space. See [Configuring licenses for team spaces on page 139](#).

You must have been assigned a role as a *Team Space Owner*. If you are a server administrator, then you are assigned with the default role of a *Team Space Owner*.

### About this task



As an owner of a team space, you can add multiple users to your **Permissive** or **Restrictive** team space.

1. Log in to Rational® Test Automation Server.

**Result**

The **Projects** page of the initial team space is displayed.

2. Click the **Settings** icon  from the left navigation pane.



**Note:** If the initial team space does not exist, then you can view the **Team Space Dashboard** page without any navigation pane.

3. Identify the team space for which you want to add a member.

You can either use the same team space where you choose to work or switch to any other team space.

4. Click **Add member** to add members in the initial team space.

Alternatively, you can select the team space, and then click **Settings > Add member**.

**Result**

The **Add Member** page is displayed.

5. Enter a username in the **Add People** field to add users.

Alternatively, you can find the username by using type search.

**Result**

The list of users is displayed.

6. Select the user.

7. Assign a role to the selected user.




**Note:** All members are assigned the default roles of a *Project Creator* and *Architect*.

You can choose the role for the user based on its associated actions:

Roles	Responsibilities
<i>Team Space Owner</i>	<ul style="list-style-type: none"> <li>◦ Create a team space.</li> <li>◦ Update and delete a team space.</li> <li>◦ Add members to a team space.</li> <li>◦ Remove members from the team space.</li> <li>◦ Assign the roles to the members.</li> <li>◦ Modify the roles of the members.</li> <li>◦ Create and modify a project.</li> <li>◦ Add or update the system model.</li> <li>◦ Add a repository to store a system model.</li> <li>◦ Update and delete a repository in a team space.</li> <li>◦ Add, update, or delete Docker.</li> </ul>

Roles	Responsibilities
	<ul style="list-style-type: none"> <li>◦ Configure or update agent and intercepts.</li> <li>◦ Register, update, delete, and list Resource Monitoring sources in a team space.</li> </ul>
<i>Project Creator</i>	<ul style="list-style-type: none"> <li>◦ Create, update, and view a project</li> <li>◦ Add, update, or delete Docker.</li> <li>◦ Configure or update agent and intercepts.</li> </ul>
<i>Architect</i>	<ul style="list-style-type: none"> <li>◦ Create, update, delete, and view a repository in a team space.</li> <li>◦ Create, update, and delete components in the system model.</li> <li>◦ Add, update, or delete Docker.</li> <li>◦ Configure or update agent and intercepts.</li> </ul>
<i>Member</i>	<ul style="list-style-type: none"> <li>◦ View the list of team spaces in the <b>Team Space Dashboard</b>.</li> <li>◦ View the configuration of the team space.</li> <li>◦ Add, update, or delete Docker.</li> <li>◦ Configure or update agent and intercepts.</li> <li>◦ View the registered Dockers, agents, or intercepts.</li> <li>◦ View the Resource Monitoring agents in a team space.</li> <li>◦ View the registered Resource Monitoring agents.</li> <li>◦ View components in the system model.</li> </ul>

**Notes:**

- The members with the assigned role of a *Team Space Owner*, *Project Creator*, *Architect*, or *Member* of a team space can be assigned with any of the following project roles:
  - *Project Owner*
  - *Tester*
  - *Viewer*
- You can add or remove the role of a member by using the **Menu** icon  next to each user in the **Members** list.

8. Repeat the applicable steps to add another user.

**Results**

You have added members with assigned roles to your team space. You can view the list of all added users on the **Team Space Configuration** page.

**What to do next**

You can reassign or remove the roles of users to one or more team spaces that you own. See [Managing members and their roles in a team space on page 612](#).

All users can request to be a member of another team space. See [Becoming a team space member on page 611](#).

Users added as members to the team space can view the team space in **My Team Spaces**.

## Modifying the configuration of a team space

After you create a team space, if you want to modify the details of the team space, add a repository to your team space, add or remove members of a team space or delete a team space, then you can perform these tasks on the **Team Space Configuration** page.

### Before you begin

You must have been assigned the role of a *Team Space Owner*.

### About this task

The **Team Space Configuration** page displays details of the team space.

As an owner of a team space, you can modify details of the team space, add members to your team space, and assign roles to the members of the team space on the **Team Space Configuration** page.

The *Team Space Owner* and *Architect* can add or remove a repository from the team space.

If the team space is no longer required, then the *Team Space Owner* can also delete that team space.


As a *Member*, *Project Creator*, or *Architect*, you can only view the configuration details of a team space on the **Details** tab of the **Team Space Configuration** page.

As a *Member*, and *Project Creator* you can only view the configuration details of a team space on the **Repository** tab of the **Team Space Configuration** page.

1. Log in to Rational® Test Automation Server.

#### Result

The **Projects** page of the initial team space is displayed.

2. Click the **Settings** icon  from the left navigation pane.
3. Identify the team space for which you want to modify the configuration.

You can either use the same team space where you are working or switch to any other team space.



4. Click **Manage team space**.

#### Result

The **Details** page is displayed.

5. Perform the modifications on the following details of the **Details** tabs of the team space:

Fields	More information
Name	Displays the name of the team space.

Fields	More information
URL alias	Displays an alias of the team space. You must suffix this alias in the application URL to access a specific team space after you log in to the application.
Available at	<p>Displays the application URL by using an alias. You can copy this URL and use it when you want to access the team space immediately after you log in to the application.</p> <p> <b>Note:</b> You cannot modify the URL.</p>
Member Access	<p>Displays the type of team space that defines the permissions to access the team space. A team space can be of the following types:</p> <ul style="list-style-type: none"> <li>◦ <b>Permissive:</b> All licensed users can access this team space without the approvals of a <i>Team Space Owner</i>.</li> <li>◦ <b>Restrictive:</b> All licensed users must get approvals from the <i>Team Space Owner</i> to access this team space.</li> </ul>
Visibility	<p>The visibility mode of a team space can be of the following types:</p> <ul style="list-style-type: none"> <li>◦ <b>Public:</b> The team spaces are visible to all licensed users in the <b>Other Team Spaces</b> panel.</li> <li>◦ <b>Private:</b> The team spaces are not visible to all the licensed users in the <b>Other Team Spaces</b> panel.</li> </ul> <p> <b>Note:</b> The visibility mode is available only when the <b>Restrictive</b> type of team space is selected.</p>
Description	Brief description of the team space. This is an optional field.
Add People	List of members in a team space.

6. Click the **Repository** tab to modify the details of the repository of the team space.

## Results

You have modified the configuration details of the selected team space.

## What to do next

You can add members and assign roles to each member of your team space.

---

Related information

[Adding members to a team space on page 604](#)

[Managing members and their roles in a team space on page 612](#)

[Adding repositories to a server project on page 620](#)

## Viewing the configuration of a team space

You can view the configuration of a team space as a member of any team space. You can view the configuration details on the **Team Space Configuration** page.


### Before you begin

You must be a member of a team space.

### About this task

The **Team Space Configuration** page displays the configuration details of the team space.

As a member of a team space, if you want to access your team space immediately after logging into IBM® Rational® Test Automation Server, then you must suffix the alias in the application URL. You can find details of the alias on the **Team Space Configuration** page. You can also view the list of members and their roles in that team space on the **Team Space Configuration** page.


1. Log in to Rational® Test Automation Server.
2. Click the **Settings** icon  from the left navigation pane.
3. Identify the team space for which you want to view the configuration.  
You can either use the same team space where you are working or switch to any other team space.
4. Click **Manage team space**.


### Result

The **Details** page is displayed.

You can view the following details about the team space:

Fields	Tabs	More information
Name	<b>Details</b>	Displays the name of the team space.
URL alias	<b>Details</b>	Displays an alias of the team space. You must suffix this alias

Fields	Tabs	More information
		in the application URL to access a specific team space after you log in to the application.
Available at	<b>Details</b>	Displays the application URL by using an alias. You can copy this URL and use it when you want to access the team space immediately after you log in to the application.
Member Access	<b>Details</b>	<p>Displays the type of team space that defines the permissions to access the team space. A team space can be of the following types:</p> <ul style="list-style-type: none"> <li>◦ <b>Permissive:</b> All licensed users can access this team space without the approvals of a <i>Team Space Owner</i>.</li> <li>◦ <b>Restrictive:</b> All licensed users must get approvals from the <i>Team Space Owner</i> to access this team space.</li> </ul>
Visibility	<b>Details</b>	<p>The visibility mode of a team space can be of the following types:</p> <ul style="list-style-type: none"> <li>◦ <b>Public:</b> The team spaces are visible to all licensed users in the <b>Other Team Spaces</b> panel.</li> <li>◦ <b>Private:</b> The team spaces are not visible to all the licensed users in the <b>Other Team Spaces</b> panel.</li> </ul> <p> <b>Note:</b> The visibility mode is available only when the</p>

Fields	Tabs	More information
		 <b>Restrictive</b> type of team space is selected.
Members	<b>Details</b>	Displays a list of members of the team space.
Repository	<b>Repository</b>	Displays the repository details that host the system model.

### Results

You have viewed the configuration details of the selected team space.

## Becoming a team space member

After logging into IBM® Rational® Test Automation Server, the licensed users must be a member of any team space to perform test activities in a project. As a licensed user, if you are not assigned the role of a member in any team space, then you can request the *Team Space Owner* to become a member of any existing team space.

### Before you begin

You must be a licensed user.

### About this task

If you want to become a member of a **Restrictive** team space, then only the owner of that team space can grant you the role of a member of a team space. If you request to become a member of a **Permissive** team space, then you automatically become a member of that team space.



**Note:** The server administrator can join any team space automatically without any approval and is assigned with a default role of *Team Space Owner*.

1. Log in to Rational® Test Automation Server.

#### Result

The **Projects** page of the initial team space is displayed.

2. Click the **Settings** icon  from the left navigation pane, and then **Join another team space**.

#### Result

The **Team Space Dashboard** is displayed.


You can view the team spaces in the following panels:

- **My Team Spaces:** You can view all the team spaces that you created. You can also view the team spaces in which you are a member.
- **Other Team Spaces:** You can view all the team spaces that are created by others.



**Note:**

You can view only the public type of **Restrictive** team spaces and the **Permissive** team spaces. You cannot view the **Restrictive** private type team spaces.

3. Select the team space that you want to join from the list of other team spaces in the **Team Space Dashboard**.
4. Click the **Key** icon  of the selected team space.
5. Confirm your request when prompted.

If you agree, then the *Team Space Owner* sees a notification about the request.

After opening the notification, the *Team Space Owner* can view the user who is requesting the access and can accept or decline the request. If the *Team Space Owner* accepts your request, you are added and assigned a *Member* role.



**Notes:**

- You can become a member of any team space only when the license of the selected team space is configured.
- If you are a server administrator, then the *Team Space Owner* role is the default role assigned.
- If you are a licensed user, then the *Member* role is the default role assigned and is restricted to specific actions.

You can then see the team space under **My Team Spaces** on the **Team Space Dashboard** page.

You might want to follow up with the *Team Space Owner* if your request is pending for some time.

## Results

You have become a member of another team space.

## What to do next

You can view the actions that you can perform based on your assigned role. See [Adding members to a team space on page 604](#).

## Managing members and their roles in a team space

After you assign roles to the members of a team space, as a *Team Space Owner*, you might want to remove a member from the team space or reassign the role to the existing members of your team space.

## Before you begin

You must have been assigned the role of a *Team Space Owner*.

## About this task




As a licensed user, you are assigned a role of a *Member* in any team space after the approval of the *Team Space Owner*.

As an owner of a team space, you can remove or reassign any member of your team space with the roles of a *Team Space Owner*, *Project Creator*, or *Architect*. You can also remove any member from the list of members of the team space.

1. Log in to Rational® Test Automation Server.

**Result**

The **Projects** page of the initial team space is displayed.

2. Click the **Settings** icon  from the left navigation pane.
3. Identify the team space for which you want to reassign the roles of the members or remove a member from the list of members.


You can either use the same team space where you are working or switch to any other team space.

4. Click **Add member** if you want to reassign the roles of the members or remove a member from the list of members of the initial team space.

Alternatively, you can select the team space, and then click **Settings > Add member**.

**Result**

The **Add member** page is displayed.

5. Identify the member from the list of **Members**.
6. Click the **Menu** icon  next to the selected member.
7. Perform any one of the following operations to the identified member of the team space:

**Choose from:**

- Click **Remove Member** to remove the identified member from the team space.
- Select the role that you want to assign or remove the assigned role for the identified member.

You can choose multiple roles for the members based on the associated actions:

Roles	Actions
<i>Team Space Owner</i>	<ul style="list-style-type: none"> <li>▪ Create a team space.</li> <li>▪ Update and delete a team space.</li> <li>▪ Add members to a team space.</li> <li>▪ Remove members from the team space.</li> <li>▪ Assign the roles to the members.</li> <li>▪ Modify the roles of the members.</li> <li>▪ Create and modify a project.</li> <li>▪ Add or update the system model.</li> <li>▪ Add a repository to store a system model.</li> <li>▪ Update and delete a repository in a team space.</li> <li>▪ Add, update, or delete Docker.</li> </ul>

Roles	Actions
	<ul style="list-style-type: none"> <li>▪ Configure or update agent and intercepts.</li> <li>▪ Register, update, delete, and list Resource Monitoring sources in a team space.</li> </ul>
<i>Project Creator</i>	<ul style="list-style-type: none"> <li>▪ Create, update, and view a project.</li> <li>▪ Add, update, or delete Docker.</li> <li>▪ Configure or update agent and intercepts.</li> </ul>
<i>Architect</i>	<ul style="list-style-type: none"> <li>▪ Create, update, delete, and view a repository in a team space.</li> <li>▪ Create, update, and delete components in the system model.</li> <li>▪ Add, update, or delete Docker.</li> <li>▪ Configure or update agent and intercepts.</li> </ul>
<i>Member</i>	<ul style="list-style-type: none"> <li>▪ View the list of team spaces in the <b>Team Space Dashboard</b>.</li> <li>▪ View the configuration of the team space.</li> <li>▪ Add, update, or delete Docker.</li> <li>▪ Configure or update agent and intercepts.</li> <li>▪ View the registered Dockers, agents, or intercepts.</li> <li>▪ View the Resource Monitoring agents in a team space.</li> <li>▪ View the registered Resource Monitoring agents.</li> <li>▪ View components in the system model.</li> </ul>



**Note:** The *Team Space Owner*, *Project Creator*, *Architect*, or *Member* of a team space can be assigned with any of the following project roles:

- *Project Owner*
- *Tester*
- *Viewer*

Similarly, you can remove the assigned role of any member of your team space.

## Results

You have completed any of the following tasks in your team space:

- Reassigned the role of the members.
- Removed the assigned role of the members.
- Removed the members.

---

Related information

[Adding members to a team space on page 604](#)

## Deleting a team space

You might want to delete a team space when it is no longer needed.

### Before you begin

You must have been assigned the role of a *Team Space Owner* of IBM® Rational® Test Automation Server.

### About this task

As a *Team Space Owner*, you can delete a team space that you do not need. When you delete a team space, it is a permanent deletion. You cannot undo the delete action on a team space.


If you delete the initial team space, then you can only view the **Team Space Dashboard** page without any navigation pane.

After you delete the team space, you can use the name of the team space again.

1. Log in to Rational® Test Automation Server.

#### Result

The **Projects** page of the initial team space is displayed.

2. Click the **Settings** icon  from the left navigation pane.
3. Identify the team space that you want to delete.

You can either use the same team space where you are working or switch to any other team space.

4. Click **Manage team space**.

#### Result

The **Details** page is displayed.

5. Click **Delete Team Space**.
6. Confirm that you want to delete the team space when prompted.

If you proceed, the team space is deleted including all of its projects and members.

### Results

You have deleted the selected team space.

### What to do next

You can view the available team spaces on the **Team Space Dashboard** page. See [Viewing team spaces on page 603](#).

After you delete a team space, the license that is used for that team space is released and is available to configure the other team spaces, if required. See [Configuring licenses for team spaces on page 139](#).

## Test assets and a server project

Rational® Test Automation Server projects manage access to your test assets, which are stored in a Git repository. Projects are maintained in a team space. Projects are either public by default or private. Private projects are not discoverable by other users. You can either add your own project or you can request to be a member of another public project.

All the licensed users are assigned with a default role of a *Project Creator*. As a *Project Creator*, when you create a project immediately after logging in the application, the project is created in the initial team space. If you want to create a project in any specific team space, then you must become a member of that team space. When you add your own project, you can configure it now or later. Configuring the project includes adding details about the project, adding one or more Git repositories, optionally adding secrets, classifying encrypted datasets, and adding a change management system. To run test assets that are associated with an encrypted dataset, you must categorize the encrypted datasets by creating a classification. You can also add users to your project to access your test assets only if the users are the members of that team space. To use Jira to create defects for the tests available in your project, you must configure the Jira server for your project on Rational® Test Automation Server.

---

### Related information

[Team space overview on page 126](#)

[Becoming a team space member on page 611](#)

[Becoming a project member on page 626](#)

[Managing access to server projects on page 627](#)

[Managing repositories on page 632](#)

[Protecting API test assets by using secrets on page 640](#)

[Managing an encrypted dataset on page 646](#)

[Default user administration on page 107](#)

[Configuring the Jira server as a change management system on page 537](#)

## Working with projects in a team space

After you log in to IBM® Rational® Test Automation Server, you must have a project to perform the test activities. To work with projects, you must create and configure projects.

### Before you begin

You must have logged into Rational® Test Automation Server.

### About this task

You must be assigned with the role of an *Owner*, *Tester*, or *Viewer* of a project.

1. Go to the **Projects** page from the **Initial Team Space** Home page. See [Viewing active projects on page 624](#).
2. Add a project. See [Adding a project on page 619](#).
3. Add a repository. See [Adding repositories to a server project on page 620](#).
4. Add secrets in secrets collections. See [Secrets configuration on page 623](#).
5. Encrypt the data set resources. See [Managing an encrypted dataset on page 646](#).
6. Configure a change management system. See [Configuration of a change management system on page 532](#).
7. Add members to a project and configure their roles. See [Adding users to a project on page 623](#).
8. Select a global branch in a project to view the test assets or resources maintained in the repositories. See [Selecting the global branch in a project on page 638](#)
9. Perform the following tasks based on the type of test assets or resources that you want to run:
  - a. Add agents to your project, if you are using remote agents as a location to run certain tests. See [Adding a performance agent to a project on page 213](#).
  - b. Add remote Dockers, if you are using a remote Docker host as a location to run tests. See [Adding a remote Docker host to the project for running tests on page 228](#).
  - c. Read the considerations that you must take into account before you configure a run or start a virtual service. See [Prerequisites for running virtual services on page 410](#).
  - d. Read the considerations that you must take into account before you configure a test run. See [Prerequisites to running tests on page 198](#).
10. Configure a run of the test asset or resource. See [Test run configurations on page 249](#).
11. View the progress of a test run. See [Viewing the progress of running test assets on page 398](#).
12. Manage a running test. See [Management of running tests on page 394](#).
13. View the results of a test run after the run is completed. See [Test results on page 455](#).
14. Review the test results and create defects for test runs from the Results page. See [Creating Jira issues for tests that ran on page 538](#).

## Results

You have performed all the tasks with the projects.

## Repository considerations for a server project

To collaborate with other project stakeholders, you can open test assets from a local clone of the Git repository, pull project test assets from a Git repository, and push changes made to your local test assets to a Git repository. Before you add a project and add a repository to that project, you must consider some information about repositories.

Consider the following sections about using Git repositories with IBM® Rational® Test Automation Server. For more information about installing, setting up, and using Git, see the [Git documentation](#).

## Git

You must install Git or upgrade to the supported version if you already have installed Git. See [System Requirements on page 12](#).

## Repositories and user identities

After you install Git, you must set up your Git repository and set up access for members. You must ensure that the repository contains your test assets.

Optionally, you can use a command line utility or Git tool to access the repository, upload your test assets, fetch or pull from the repository, push to the repository, clone the repository, and other operations you want to perform in Git.

## Local and shared repositories

After you create a remote or shared repository in Git, you can create a local version of the repository by cloning the remote repository. You must ensure that your test assets are available in the remote repository and are also cloned to the local repository.

Alternatively, if your test assets are on your local system, you can set up a Git repository in the bare mode, add the project files to the local repository, and then commit and push from the local repository to the remote repository in Git by using your preferred method.



**Note:** While copying the test assets from your local system to the repository, you must ensure that you copy the entire project that contains the test assets.

## User authentication for the Git repository

The administrator can set up different types of authentication for accessing the Git repository. Rational® Test Automation Server supports the following authentication types:

- HTTP with username and password
- HTTP without username and password
- HTTPS with username and password
- HTTPS without username and password
- SSH with SSH key and passphrase
- SSH with SSH key and without a passphrase

Based on the authentication type that is set for a repository, you must provide the same authentication values in Rational® Test Automation Server when you add a repository.

## Test assets

You must complete the following tasks in the desktop client where you are authoring your test before you check in and commit the test assets to the Git repository.

Test type	Task	More information
API suite in Rational® Integration Tester	Change the local stub to a remote stub.	See <a href="#">Test run considerations for API Suites or API tests on page 200</a> .
	Add the library files.	

## Adding a project

The first step is to add a project and provide some details about it.

### Before you begin

To add one or more projects to manage access to your test assets, you must log in to IBM® Rational® Test Automation Server by providing the application URL in a browser.

If you are a new user, and LDAP and Active Directory are not configured, then you must first sign up by completing a form that specifies user information such as an email, user name, and password. You can then log in by using that information.

You must be a member of any licensed team space.

### About this task

As a licensed user, you are by default assigned as both *Project Creator* and *Architect* role for the initial team space. You must select a team space to create a project.


You can view all the team spaces in the **Team Space Dashboard**. To create a project in any team space, you must be a member of that team space and assigned a *Team Space Owner* or *Project Creator* role in that team space.


After you select a team space from the **Team Space Dashboard**, you can add a project and give it a name and description.

1. Log in to Rational® Test Automation Server.

#### Result

The **Projects** page of the initial team space is displayed.

2. Click **New project** or  icon from the navigation pane to create a project in the team space where you are working.

Alternatively, if you want to create a project in any other team space, then switch to another team space from the **Settings** icon  in the navigation pane, and then create a project.

#### Result

The **Details** page is displayed.

### Results

You have successfully created a project.

**What to do next**

You can modify your project configuration. You can also add a repository to your project.

---

 Related information

[Default user administration on page 107](#)

[Changing the visibility configuration of a project on page 625](#)

[Adding repositories to a server project on page 620](#)

**Adding repositories to a server project**

You can add a single repository individually or multiple repositories simultaneously to a server project to access assets and resources that are available in the repositories.

**Before you begin**

You must have completed the following tasks:

- Added a project on IBM® Rational® Test Automation Server.
- Been granted permission and you must have set up your credentials to access the remote repositories in a repository domain.

**About this task**

As a project owner or a tester, you can add a single repository or multiple repositories to your project. You can use the information in the following table to decide the method for adding repositories to your project:

When...	Action...
You want to add any of the following repositories: <ul style="list-style-type: none"> <li>• A single repository owned by you.</li> <li>• A single repository owned by other people.</li> <li>• The repositories that are in different domains.</li> </ul>	You can add individual repositories one at a time.  You must then provide the complete URL of the repository along with the required credentials to authenticate the connection to the repository from Rational® Test Automation Server.  For example, the URL to a single repository can be as <code>https://github.com/&lt;user&gt;/&lt;repository&gt;.git</code> . You must provide the <i>username</i> and <i>password</i> to access this repository.
You want to add any of the following repositories:	You can add all the repositories simultaneously that are in the same domain.



When...	Action...
<ul style="list-style-type: none"> <li>• Multiple repositories that are owned by you and are contained in the same domain.</li> <li>• Multiple repositories that are owned by others and are contained in the same domain.</li> </ul>	<p>You must then provide the URL of the repository domain along with the required credentials to authenticate the connection to the repository from Rational® Test Automation Server.</p> <p>For example, the URL of the repository domain can be as <code>https://github.com</code>. You must select the platform and provide your <i>personal access token</i> to access the repositories.</p>

1. Log in to Rational® Test Automation Server and open the team space that contains your project.
2. Open your project either from the navigation pane or from the **My Projects** page.

The **Overview** page of the project is displayed.

3. Click **Manage > Configuration** from the navigation pane.
4. Click the **REPOSITORIES** tab.
5. Click **Add repository**.

The **Add repository** dialog is displayed.

You can either choose to add a single repository individually or multiple repositories simultaneously from a single domain.

6. Perform either of the following actions:
  - To add a single repository, go to [Step 7 on page 621](#).
  - To add multiple repositories from a domain, go to [Step 8 on page 622](#).
7. Perform the following steps to add individual repositories at a time:

- a. Enter the complete URL of the repository in the **Git URL** field.

For example, the URL to a single repository can be as `https://github.com/<user>/<repository>.git`.

The required authentication fields are displayed that depend on the protocol you specified in the URL.

If...	Then...
The repository URL contains <i>http</i> or <i>https</i>	The following fields for authentication are displayed: <ul style="list-style-type: none"> <li>▪ <b>Username</b></li> <li>▪ <b>Password</b></li> </ul>
The repository URL contains <i>git@</i>	The following fields for authentication are displayed:

If...	Then...
	<ul style="list-style-type: none"> <li>▪ SSH/Deploy Key</li> <li>▪ Passphrase</li> </ul>



**Note:** If you use SSH to connect to a remote repository and Rational® Test Automation Server displays an `Auth Fail` error, you can resolve this error by regenerating your SSH keys by using the **-m PEM** option.

- b. Enter the credentials to authenticate the connection to the repository.
- c. Click **Add** to add the repository to your project.

You can add other individual repositories by specifying the URL and performing the actions in [Step 7 on page 621](#).

8. Perform the following steps to add multiple repositories simultaneously that are contained in the same domain:

- a. Enter the URL of the repository domain in the **Git URL** field.

For example, the URL of the repository domain can be as `https://github.com`.

- b. Select the platform from the list under **Platform** where your repositories reside.
- c. Enter your *personal access token* to authenticate your connection to the repository domain.



**Note:** If you do not have your *personal access token*, you can click **Generate** to generate your *personal access token* and use the token to authenticate your connection.

- d. Click **Next**.

The connection to the repository domain is established and all the repositories that you own are listed. The time when the repository was last updated is displayed in the **Last updated** column.

- e. Select the repositories that you want to add to your project.
- f. Click **Add**.



**Note:** Depending on the size of the repository that you are adding, it can take a few to several minutes to add the repository to your project on the server.

All the repositories that you selected are added to your project and are listed on the **Repositories** page.

## Results

You have added individual repositories or multiple repositories to your project on the server.

**What to do next**

You can perform the following actions on the repository that you added:

- Update the authentication credentials if they are changed in the Git repository configuration. See [Updating the authentication credentials of the repository on page 635](#).
- Delete a repository if it is no longer required. See [Deleting a repository on page 636](#).
- Refresh a repository to fetch and synchronize changes from the remote repository. See [Refreshing repositories manually on page 636](#).
- Configure a webhook to notify you when there is a push event in the remote repository. See [Creating webhooks on page 639](#).

**Secrets configuration**

If you are working with an API suite and the project test asset contains environment variables that are required for the test runs, you must configure the environment variables as secrets in a secrets collection by using the **Secrets** tab. Configuring secrets enable the API suite to run correctly on Rational® Test Automation Server.

---

Related information

[Protecting API test assets by using secrets on page 640](#)

**Adding users to a project**

You can add users to your server project so they can access your test assets.

**Before you begin**

- You own a project.
- The users you want to add to your project must be the members of the team space of that project.


**About this task**

As a project owner, you can add other users to your public or private project. For example, if you have a large number of test assets that must be run, you might want your test team to help run them, and see the results.

1. Log in to Rational® Test Automation Server.

**Result**

The **Projects** page of the Initial Team Space is displayed.

2. Select the project for which you want to add the users.
3. Click the **Settings** icon  on the Project card.
4. Add users by entering the username of the user you want to add. You can add a partial name and then press the **Enter** key to see the user that you want to add.
5. Select that user and then assign a role.
6. Click **Viewer**, **Tester**, or **Owner**. When you add a user, the default role is *Viewer*.

The added user after logging in can see the owner's project in their list of **My Projects**.

7. Repeat the applicable steps to add another user.

### Results

You have successfully added users to your project.

### What to do next

You can add or remove roles to one or more projects that you own. All users can request to be a member of another public project.

---

Related information

[Becoming a project member on page 626](#)

[Managing access to server projects on page 627](#)

## Viewing active projects

After you log in to IBM® Rational® Test Automation Server, you can view both active and archived projects in the **Initial team space**.

### About this task

As a project owner, you can view your projects under the **My Projects** panel, and the projects that are created by other users as public projects are displayed under the **Other Projects** panel. You can view a list of all active projects that you created under **My Projects**. You can view a list of all archived projects under **Archived Projects**.

Log in to Rational® Test Automation Server.

The **Projects** page is displayed.

You can view the following projects:

- The active projects that you created or added to as a member with either the *Tester* or *Viewer* role are displayed under the **My Projects** panel.
- The active projects that are created by others as public projects are displayed under the **Other Projects** panel.

### Results

You have viewed the active projects that are created by you and other users in the team space.

### What to do next

You can perform any of the following tasks:

- Create a project. See [Adding details to a server project on page 619](#).
- Request to become a member of a project that is displayed under the **Other Projects** panel. See [Becoming a project member on page 626](#).
- View the other pages in the team space and perform tasks on those pages. See [Tasks in a team space on page 131](#).

---

#### Related information

[Archiving or unarchiving server projects on page 629](#)

## Changing the visibility configuration of a project

After you create a project, you might want to change the visibility configuration of the project from the public, which is the default, to private.

### Before you begin

You must have created a project.

1. Log in to Rational® Test Automation Server.

#### Result

The **Projects** page of the initial team space is displayed.

2. Select the project from the left navigation pane.

#### Result

The project page is displayed.

3. Click **Manage > Configuration >** .

Alternatively, select a project, and then click the **Configure project** icon  on the project card.

#### Result

The **Details** page is displayed.

4. Select **Private** as the visibility mode of the project.
5. Click **Save**.

### Results

You have successfully changed the visibility configuration of a project.

### What to do next

You can add a repository to your project.

---

#### Related information

[Default user administration on page 107](#)

## Becoming a project member


You might want to request to be a member of another project. New users without any projects might also want to be a member of an existing project.

### About this task



As a project owner, you can add users to your public or private projects. All users can request to become a member of a public project.




**Note:** You can add a licensed user as a member of your project only if that user is the member of your team space.

1. Request to be a member of another project. Search for the project that you want to join in the list of other projects from the project Home page. Click that Project card or the **Key** icon . Only public projects are visible in the list of projects.
2. Confirm your request when prompted. If you agree, the project owner is notified. The project owner sees a pending request **Notification** icon on their Project card.

### My Projects

 myproj


No description.

Owner: [user55](#) 

Created: [a month ago](#)

Branch: **master**

Test Suites: **59**    Individual Tests: **40**

Last Run Time: Nov 4, 2020 8:59 AM

---

from beginning

1	0/59	1/59
In Progress	Fail	Success

After opening the notification, the owner can see the user that is requesting access and can accept or decline it.

If the project owner accepts your request, you are added as a member of the project with a Viewer role. A Viewer role is the default role assigned and is restricted to specific actions.

You then see the project under My Projects on the Home page.

If the project owner declines your request, follow up with the project owner.

**Results**

You have successfully sent a request to become a member of another project.

**Managing access to server projects**

You might want to add or remove a project member role from your project.

**About this task**

As a project owner, you can assign access by specifying a role when you add a user to your project. You can also assign more access or remove access for a user. Roles enable users to perform tasks on project resources such as running tests and viewing test results.

1. Choose the role in a project based on its associated actions.

Option	Description
<p><b>Owner of a project</b></p>	<ul style="list-style-type: none"> <li>◦ All Tester's actions</li> <li>◦ Update, archive, list, and view a project</li> <li>◦ Assign, update, delete, and list user roles for a project</li> <li>◦ Accept new member requests</li> <li>◦ Create, update, and delete a dataset classification in a project</li> <li>◦ Add, update, and remove an encrypted dataset from a classification</li> <li>◦ Update current row for a listed dataset in a project</li> <li>◦ Delete, list and view a report</li> <li>◦ Update, delete, and list a repository in a project</li> <li>◦ Get the test assets from a repository in a project</li> <li>◦ Create, update, delete and list secrets in a project</li> <li>◦ Create, update, delete, and list resource monitoring sources in a project</li> <li>◦ Get the content of a resource monitoring source in a project</li> <li>◦ Register, update, delete, and list resource monitoring agents</li> <li>◦ Create, configure, and delete the Jira change management system</li> </ul>

Option	Description
	<ul style="list-style-type: none"> <li>◦ Create and update defects in the Jira change management system</li> <li>◦ Create and update components in the system model</li> </ul>
<b>Tester of a project</b>	<ul style="list-style-type: none"> <li>◦ All Viewer's actions</li> <li>◦ List repositories in a project</li> <li>◦ Run a test and create a report</li> <li>◦ Stop a test while running</li> <li>◦ Get the test assets from a repository in a project</li> <li>◦ Get a list of datasets in a project</li> <li>◦ Get the content of a dataset</li> <li>◦ Update the current row for a listed dataset in a project</li> <li>◦ Add, update, and remove an encrypted dataset from a classification</li> <li>◦ Delete a report if you created the report</li> <li>◦ Create and update defects in the Jira change management system</li> <li>◦ Create and update components in the system model</li> </ul>
<b>Viewer of a project</b>	<ul style="list-style-type: none"> <li>◦ List your own roles on a project</li> <li>◦ List running tests</li> <li>◦ List and view a report</li> <li>◦ List resource monitoring sources in a project</li> <li>◦ Get the content of a resource monitoring source in a project</li> <li>◦ List the resource monitoring agents</li> <li>◦ View components in the system model</li> <li>◦ View defects in the Jira change management system</li> </ul>
<b>Non-member of a project</b>	<ul style="list-style-type: none"> <li>◦ Request to be a member of a project</li> </ul>

2. Open the project that you own.

From the project details, you can see a list of project members for your project and then add or remove a members role.

3. Add or remove the member access by using the **Menu** icon  next to each user in the member list.



## Archiving or unarchiving server projects

You can archive a project when that project is no longer needed but you want to retain it for future reference. If you do want to use an archived project, you can unarchive it.


### About this task

As a project owner, you can archive inactive projects. Archived projects are not visible to users or project members, but owners can show or hide their archived projects. You can view the list of all archived projects from the navigation pane. Archived projects can be unarchived.



#### Note:

- To archive a project:

1. Log in to Rational® Test Automation Server.
2. Select your project from the **Projects** page.
3. Click the **Configure project** icon  on the project card.

#### Result

The **Details** page is displayed.

4. Click **Archive Project**.
5. Confirm that you want to archive the project when prompted.

If you proceed, the project is archived and is hidden from **My Projects**.


- To view an archived project:

1. Log in to Rational® Test Automation Server.
2. Select your team space.
3. Click **Archived Projects**.

#### Result

A list of all archived projects that are associated with you is displayed under **My Projects** and the archived projects that are associated with others are displayed under **Other Projects**.

- To unarchive a project:

1. Log in to Rational® Test Automation Server.
2. Click **Archived Projects**.
3. Select your archived project, and then click the **Configure project** icon  on the project card.

#### Result

The **Details** page is displayed.

4. Click **Unarchive Project**.
5. Confirm that you want to unarchive the project when prompted.  
If you proceed, the project is unarchived and is shown under **My Projects**.

## Deleting server projects

You might want to delete a project when the project is no longer needed, and you want to free up disk space.

### About this task

As a project owner, you can delete a project that you no longer want. When you delete a project it is permanent. You cannot undo the delete project action.

After you delete the project, you can use the project name again.

1. Open the project that you want to delete.
2. From the project details, delete the project.
3. Confirm that you want to delete the project when prompted. If you proceed, the project is deleted including all of its test assets, results, and members.



**Note:** The repository that you used for the project is not deleted.

## An overview of test assets, modifications, and scheduled runs

After you configure a project with one or more Git repositories and run some test assets, you can view statistics about those test assets.

From the **Overview** page, several cards display charts, sliders, and tables of the statistics of the test assets in your server project. You can click some of the charts, sliders, or tables to see more information, such as the progress of a scheduled run or the results of a run. You can also review some statistics by date and see more details such as the proportion of test assets by type.

You can view a server project, which might contain multiple repositories and multiple branches, by selecting the name of the branch from the **Branch** drop-down list. For more information about selecting branches, see [Selecting the global branch in a project on page 638](#).

The **Members** area displays the total number of users who have access to the project. You can click the number to view the details of all the members. The **Owner** area displays the username of the project owner.

### Test suites

This card displays a slider of the percentage of test suites by type. A test suite is a collection of tests, including schedules, compound tests, API Suites, and AFT Suites. You can hover over the slider and see the proportion of test suites by test suite type. The number of suites shown are relative to the selected branch.

### Individual tests

This card displays a slider of the percentage of individual tests by type. You might have performances tests, API tests, or UI tests according to the desktop clients that you used to create the tests. You can hover over the slider and see the proportion of individual tests by type. The number of tests shown are relative to the selected branch.

### Execution results

This card displays a chart of the proportion of test suites that were run by status and by date. If no date is selected, the card displays all the run results associated with the project since it was created. You can see the run results by clicking a test or Suite status. You can also hover over the chart and see the test or Suite status and the proportion of test assets. You can view statistics by tests or Suites that ran.



#### Notes:

- The results shown are generated from tests or Suites run.
- The statistics exclude canceled runs of tests or Suites.
- The number of Suites and individual tests that are associated with your project are shown on the **Home** page along with a verdict summary. You can see the run results by clicking a verdict. You can also see the last date and time when you ran one or more test assets.
- The number of suites shown are relative to the selected branch.

### Last run

This card displays a table of the last three runs by date and time. You can see the run results by clicking each last run.

### New or modified

This card displays a chart of new and modified test assets that were made in your Git repository for a week or for the last seven days based on a selected date range. You can hover your mouse over each bar in the chart and see the proportion of test assets by type. The data shown is relative to the selected branch.

### Scheduled runs

This card displays a table of the next three scheduled runs by date and time. You can see the progress of a scheduled run by clicking each scheduled run.

---

#### Related information

[Test assets and a server project on page 616](#)

## Managing repositories

After you create a project in Rational® Test Automation Server, you can add repositories that contain test assets and resources to the project. You can also change the repository details, update the user credentials of a repository, delete a repository, create a webhook for the configured repository.

### Tasks in a repository

To add a repository in your project and work on the repository, you can perform the tasks listed in the following table:

Task	More information
Adding a repository	<a href="#">Adding repositories to a server project on page 620</a>
Updating the repository credentials	<a href="#">Updating the authentication credentials of the repository on page 635</a>
Refreshing a repository	<a href="#">Refreshing repositories manually on page 636</a>
Creating a webhook for the repository	<a href="#">Creating webhooks on page 639</a>
Selecting the global branch	<a href="#">Selecting the global branch in a project on page 638</a>
Deleting a repository	<a href="#">Deleting a repository on page 636</a>

### Adding repositories to a server project

You can add a single repository individually or multiple repositories simultaneously to a server project to access assets and resources that are available in the repositories.

#### Before you begin

You must have completed the following tasks:

- Added a project on IBM® Rational® Test Automation Server.
- Been granted permission and you must have set up your credentials to access the remote repositories in a repository domain.

#### About this task

As a project owner or a tester, you can add a single repository or multiple repositories to your project. You can use the information in the following table to decide the method for adding repositories to your project:

When...	Action...
<p>You want to add any of the following repositories:</p> <ul style="list-style-type: none"> <li>• A single repository owned by you.</li> <li>• A single repository owned by other people.</li> <li>• The repositories that are in different domains.</li> </ul>	<p>You can add individual repositories one at a time.</p> <p>You must then provide the complete URL of the repository along with the required credentials to authenticate</p>

When...	Action...
	<p>the connection to the repository from Rational® Test Automation Server.</p> <p>For example, the URL to a single repository can be as <code>https://github.com/&lt;user&gt;/&lt;repository&gt;.git</code>. You must provide the <i>username</i> and <i>password</i> to access this repository.</p>
<p>You want to add any of the following repositories:</p> <ul style="list-style-type: none"> <li>• Multiple repositories that are owned by you and are contained in the same domain.</li> <li>• Multiple repositories that are owned by others and are contained in the same domain.</li> </ul>	<p>You can add all the repositories simultaneously that are in the same domain.</p> <p>You must then provide the URL of the repository domain along with the required credentials to authenticate the connection to the repository from Rational® Test Automation Server.</p> <p>For example, the URL of the repository domain can be as <code>https://github.com</code>. You must select the platform and provide your <i>personal access token</i> to access the repositories.</p>

1. Log in to Rational® Test Automation Server and open the team space that contains your project.
2. Open your project either from the navigation pane or from the **My Projects** page.

The **Overview** page of the project is displayed.

3. Click **Manage > Configuration** from the navigation pane.
4. Click the **REPOSITORIES** tab.
5. Click **Add repository**.

The **Add repository** dialog is displayed.

You can either choose to add a single repository individually or multiple repositories simultaneously from a single domain.

6. Perform either of the following actions:
  - To add a single repository, go to [Step 7 on page 633](#).
  - To add multiple repositories from a domain, go to [Step 8 on page 634](#).
7. Perform the following steps to add individual repositories at a time:

- a. Enter the complete URL of the repository in the **Git URL** field.

For example, the URL to a single repository can be as `https://github.com/<user>/<repository>.git`.

The required authentication fields are displayed that depend on the protocol you specified in the URL.

If...	Then...
The repository URL contains <i>http</i> or <i>https</i>	The following fields for authentication are displayed: <ul style="list-style-type: none"> <li>▪ <b>Username</b></li> <li>▪ <b>Password</b></li> </ul>
The repository URL contains <i>git@</i>	The following fields for authentication are displayed: <ul style="list-style-type: none"> <li>▪ <b>SSH/Deploy Key</b></li> <li>▪ <b>Passphrase</b></li> </ul>



**Note:** If you use SSH to connect to a remote repository and Rational® Test Automation Server displays an `Auth Fail` error, you can resolve this error by regenerating your SSH keys by using the **-m PEM** option.

- b. Enter the credentials to authenticate the connection to the repository.
- c. Click **Add** to add the repository to your project.

You can add other individual repositories by specifying the URL and performing the actions in [Step 7 on page 633](#).

8. Perform the following steps to add multiple repositories simultaneously that are contained in the same domain:

- a. Enter the URL of the repository domain in the **Git URL** field.

For example, the URL of the repository domain can be as `https://github.com`.

- b. Select the platform from the list under **Platform** where your repositories reside.
- c. Enter your *personal access token* to authenticate your connection to the repository domain.



**Note:** If you do not have your *personal access token*, you can click **Generate** to generate your *personal access token* and use the token to authenticate your connection.

- d. Click **Next**.

The connection to the repository domain is established and all the repositories that you own are listed. The time when the repository was last updated is displayed in the **Last updated** column.

- e. Select the repositories that you want to add to your project.
- f. Click **Add**.



**Note:** Depending on the size of the repository that you are adding, it can take a few to several minutes to add the repository to your project on the server.

All the repositories that you selected are added to your project and are listed on the **Repositories** page.

## Results

You have added individual repositories or multiple repositories to your project on the server.

## What to do next

You can perform the following actions on the repository that you added:

- Update the authentication credentials if they are changed in the Git repository configuration. See [Updating the authentication credentials of the repository on page 635](#).
- Delete a repository if it is no longer required. See [Deleting a repository on page 636](#).
- Refresh a repository to fetch and synchronize changes from the remote repository. See [Refreshing repositories manually on page 636](#).
- Configure a webhook to notify you when there is a push event in the remote repository. See [Creating webhooks on page 639](#).

## Updating the authentication credentials of the repository

If the authentication credentials of the Git repository which was added to your project are changed, then you must update the credentials of the repository in Rational® Test Automation Server.

### Before you begin

You must have completed the following tasks:

- Been assigned a *Member* or *Project Creator* role in a team space.
- Been assigned the *Tester* or *Owner* role in the project with access permissions to edit repositories.

### About this task

In Rational® Test Automation Server, the repository is refreshed at regular intervals to pull the latest changes committed to the Git repository. After you update your authentication credentials of the repository, the repository is refreshed to ensure that you have access to the Git repository and to the latest assets in the repository.

1. Log in to Rational® Test Automation Server and open the team space that contains your project.
2. Open your project either from the left navigation pane or from the **My Projects** page.
3. Click **Manage** from the left navigation pane.

The **Repositories** page of the project is displayed.

4. Click the **Open action menu** icon inline with the repository that you want to update, and then click **Change...**

The **Edit Repository** dialog is displayed.

5. Enter the new credentials, and then click **Update**.

The new credentials are updated and your access to the repository is retained.

## Deleting a repository

You can delete repositories that you no longer require in your test environment.

### Before you begin

You must have completed the following tasks:

- Configured the repository that contains the test assets in your project.
- Been assigned a *Owner* or a *Tester* role with access to delete repositories.

### About this task

You can delete a repository that you have configured for your project.

1. Log in to Rational® Test Automation Server and open the team space that contains your project.
2. Open your project either from the left navigation pane or from the **My Projects** page.
3. Click **Manage** from the left navigation pane.

The **Repositories** page of the project is displayed.

4. Click the **Open action menu** icon inline with the repository that you want to update, and then click **Delete**.

The **Delete repository** dialog is displayed.

5. Select the checkbox, and then click the **Delete** button.

The repository is deleted and removed from the list.

## Refreshing repositories manually

You can manually trigger the refresh activity for a repository to instantly fetch and synchronize the changes from the remote repository.

### Before you begin

You must have completed the following tasks:

- Configured the repository that contains the test assets in your project.
- Been assigned a *Member* or a *Project Creator* role.
- Been assigned a *Owner* or a *Tester* role.




**About this task**

Every two minutes, the repository is refreshed automatically. You can also manually trigger the refresh activity for a repository.

1. Log in to Rational® Test Automation Server and open the team space that contains your project.
2. Open your project either from the left navigation pane or from the **My Projects** page.
3. Click **Manage** from the left navigation pane.

The **Repositories** page of the project is displayed.





4. Click the **Open action menu** icon  inline with the repository, and then select **Refresh**.

The refresh activity is triggered.



**Note:** The **Refresh** option is disabled when the refresh is in progress. After the refresh is completed, the **Refresh** option is enabled.

The following table lists the icons and their description based on the status of the refresh activity:

Refresh icon	Description
	The last refresh activity was successful.
	<p>The last refresh activity was successful but errors were found while processing some of the test assets. To view the error details of the test assets for a specific repository, you can click the arrow inline with the repository name.</p> <p>You can resolve the errors of these test assets in the Git server, and then refresh the server repository, if required.</p> <p> <b>Note:</b> The repository, which contains the branch that is selected in the <b>Branch</b> field, displays the unprocessed test assets for that branch. Other repositories display the unprocessed test assets for their default branch.</p>
	<p>The refresh activity was not successful due to any one of the following reasons:</p> <ul style="list-style-type: none"> <li>◦ Internal error</li> <li>◦ Connection issue</li> <li>◦ Authentication failure</li> <li>◦ Disk quota exceeded</li> <li>◦ Repository unreachable</li> </ul>

## Results

You have manually refreshed the repository and the **Last Refreshed** field is updated with the following details:

- The time when it was last refreshed in the remote repository.
- Latest commit ID and message of the remote repository.

## Selecting the global branch in a project

After you add one or more repositories to your project, you can start accessing all branches in all repositories of that project. You can select only one branch as a global branch from the list of branches in the **Execution**, **Datasets**, and **Overview** pages.

### Before you begin

You must have completed the following tasks:

- Been assigned a *Member* or *Project Creator* role in a team space.
- Added at least one repository in your project.

### About this task

You can access all the branches contained in all the repositories, which are added to the project, by using the **Branch** field on the **Execution**, **Datasets**, and **Overview** pages.

After you select a branch from any of the repositories as the global branch, this selection is retained for you as the global branch unless you change the branch on any of the following pages: **Overview**, **Execution**, or **Datasets** pages.



#### Note:

- When you add the first repository to a project, the global branch is set to the default branch of the repository.
- Branches are arranged in an increasing order of their names in the list.
- When you change the branch in one of the pages, then the same branch is displayed as selected in the other pages and the contents of that page displays the assets from the selected branch.

1. Log in to Rational® Test Automation Server and open the team space that contains your project.
2. Open your project either from the left navigation pane or from the **My Projects** page.

The project **Overview** page is displayed.

3. Select a branch from the **Branch** field.

The branch that you select on any of the following pages is reflected in the rest of the other pages and the test assets are filtered based on the global branch selection on each of these pages.:

- **Overview**
- **Execution**
- **Datasets**

## Results

You have selected a global branch in a project.


## Creating webhooks

You can configure a webhook for a repository in the Git server to notify Rational® Test Automation Server whenever there is a push event in the remote repository. Then, the refresh activity is triggered to fetch the latest changes to the server repository.

### Before you begin

You must have completed the following tasks:

- Configured the repository that contains the test assets in your project.
- Been assigned a *Member* or *Project Creator* role in a team space.
- Been assigned the *Tester* or *Owner* role in the project.

1. Log in to Rational® Test Automation Server and open the team space that contains your project.
2. Open your project.
3. Go to the **Repositories** page.
4. Click the **Menu** icon  and then click **Webhook**.

The **Webhook Configuration** dialog is displayed.

5. Copy the URL and the Secret of the repository that you want to configure.



**Note:** The URL and the Secret details are specific to a repository.

6. Click the link of the repository for which you want to configure the webhook.

The remote repository page is displayed.

7. Paste the details, which you copied from Rational® Test Automation Server, in the appropriate fields of the remote repository.

## Results

You have configured the webhook for a repository in the Git server. The changes of the remote repository are synchronized in the repository of your project.

## Protecting API test assets by using secrets

Secrets are key-value pairs that are created for your project in IBM® Rational® Test Automation Server under a secrets collection. You can create secrets collections for your project that enable you or members in your project to use secrets at test runtime either in Rational® Test Automation Server or in desktop clients.

The secrets collections in a project in Rational® Test Automation Server has a separate access control list managed by the members with access to the secrets collections. Controlling access to secrets means controlling access to applications and systems under test. The introduction of secrets (under secrets collections) for a project has simplified managing access to separate environments. If a member of a project does not have access to a secret, for example, a *server credential* then the member cannot accidentally or maliciously run tests against that server. For example, tests that must access the database server by using the server credentials to retrieve stored data can only be run by a member if the access to the secrets is granted.



**Note:** Secrets and secrets collections are applicable to test assets authored in IBM® Rational® Integration Tester that enable running tests in defined environments. Secrets are not applicable to tests authored in Rational® Functional Tester or Rational® Performance Tester.

As a project member with the *Owner* or *Tester* role, you can create secrets collections in the project. You can grant or restrict access to the secrets collection that you create in the project.

Members with access to a secrets collection can access, edit, or delete the secrets collection in Rational® Test Automation Server and can view secrets, edit secrets, or delete secrets.

Members with access to secrets collections can grant access to or remove the following:

- Other members added specifically
- All members with a specific role

Members in the project with the *Owner* or *Tester* role and with access the secrets collection can use the secrets in the secrets collection, in tests at runtime.

If you are configuring a project to run an API Suite with tests that refer to secret values, you must configure the secrets under a secrets collection by using the **SECRETS** tab. You must complete the following tasks:

1. Create a secrets collection. See [Step 1 on page 641](#) in [Managing secrets collections on page 640](#).
2. Add secrets in the secrets collection created. See [Step 1 on page 643](#) in [Creating a secret in a secrets collection on page 642](#).
3. Grant access to project members or member roles, who can access the secrets collection. See [Step 1 on page 644](#) in [Granting access to members or member roles on page 643](#).

## Managing secrets collections

You can create secrets in a secrets collection for your project. Secrets are credentials required in certain tests during test runs. Secrets stored in the collection can be used by members to run tests on different environments and

eliminates the need to store secrets in multiple locations. You can opt to edit or delete a secrets collection that you configured for your project any time after you create a secrets collection.

### Before you begin

- You must have created a project on Rational® Test Automation Server. See [Test assets and a server project on page 616](#).
- You must have completed the following tasks before you edit or delete a secrets collection:
  - Configured a secrets collection in your project.
  - Created secrets in the selected secrets collection. See [Creating a secret in a secrets collection on page 642](#).
- You must be a member with the *Owner* or *Tester* role to create a secrets collection.
- You must be a member with access to the secrets collection to edit or delete the secrets collection. See [Granting access to members or member roles on page 643](#).

### About this task


You must configure secrets collections in your project so that the members of the project can use secrets contained in a collection during test runs. You can configure secrets so that you can use them in different test environments.

As a member with access to the secrets collection, you can opt to edit or delete a secrets collection configured in a project. For example, you might want to edit the secrets collection name or delete the secrets collection if the testing environment has changed and if secrets that are configured earlier are not required.

- To create a secrets collection, go to [Step 1 on page 641](#).
- To edit or delete a secrets collection, go to [Step 4 on page 642](#).

To create a secrets collection:

1. To create a secrets collection while configuring a new project in the Rational® Test Automation Server UI, open the **SECRETS** tab in the **Project Configuration** and create a secrets collection. Use **Add Collection**.
2. Alternatively, to create a secrets collection in an existing project, complete the following steps:
  - a. Log in to Rational® Test Automation Server and from the User Interface (UI) open the project listed under **My Projects** for which you want to create a secrets collection.
  - b. Open the Project Configuration page, and then open the **SECRETS** tab to create a secrets collection.
3. Enter a name for the secrets collection as its *Identifier*.

 **Tip:** You can create a secrets collection that contains secrets for a particular test environment in your project. For example, the secrets collection *test\_env* can contain secrets that application testers can use in tests that they run while the secrets collection *dev\_env* can contain secrets that application developers can use in tests they run.

A message is displayed for the successful creation of the secrets collection.

The secrets collection created is displayed.

You can add secrets to the secrets collection you created.

To edit or delete a secrets collection:


4. Log in to Rational® Test Automation Server and from the UI open the project listed under **My Projects**.

5. Open the secrets collection from the **SECRETS** tab in the Project Configuration page.

If there are multiple secrets collections in the project, select the secrets collection that you want from the list.

- To edit a secrets collection, go to [Step 6 on page 642](#).
- To delete a secrets collection, go to [Step 7 on page 642](#).

6. To edit a secrets collection, complete the following steps:

a. Click the **Edit** icon  to edit the selected secrets collection.



**Note:** The **Edit** icon  is displayed only for the project owner.

b. Edit the name of the secrets collection, and update the secrets collection.

The secrets collection is updated with the updated name.

7. To delete a secrets collection, click the **Delete** icon  to delete the selected secrets collection.

The selected secrets collection is removed from the list of secrets collections configured for the project.

## Results

You have completed the following tasks:

- Created a secrets collection for your project.
- Edited the name of a secrets collection in your project.
- Removed a secrets collection from your project.

## What to do next

- If you have created a new secrets collection, you must add secrets to your secrets collection.
- You must provide access to project members or member roles to the secrets collection by selecting members or member roles.

## Creating a secret in a secrets collection

You must create secrets in the secrets collections configured in your project so that the secrets contained in a collection can be used in certain tests by members of the project with access to the secrets collections during an API suite run.

**Before you begin**

You must have created a project on Rational® Test Automation Server and configured a secrets collection in your project.

You must be a member with access to the secrets collection.

**About this task**

You can also configure secrets such that the secrets can be used across different test environments by members with access to the secrets collection. Secrets correspond to the environment variables or tags that you create in a Rational® Integration Tester project specific to an environment.

1. To create a secret under a secrets collection while configuring a new project in the Rational® Test Automation Server UI, select the secrets collection listed in the **SECRETS** tab in the Project Configuration page and create a secret under the secrets collection.
2. Alternatively, to create a secret under a secrets collection in an existing project, complete the following tasks:
  - a. Log in to Rational® Test Automation Server and from the UI open the project listed under **My Projects**.
  - b. Open the secrets collection from the **SECRETS** tab in the Project Configuration page.
3. Enter a name for the secret as its *Identifier* and enter the `password` as the *Value* for the secret. For example, under the secrets collection (named as `test_env`), enter the name of the secret to access a database as `dbcred` and enter the `password` required to access the database as its value.

A message is displayed for successful creation of the secret.

**Results**

You have created secrets in the selected secrets collection for your project.

**What to do next**

- You can view, edit, or delete the secrets created under a secrets collection any time you want.
- You can use the secrets in the tests that require these secrets during test runs.

**Granting access to members or member roles**

You can grant or revoke access to the secrets collection in your project to individual members with different roles or the all members with a specific role. Without access to the secrets collection, members cannot view, create, edit, delete, or use the secrets in the secrets collection.

**Before you begin**

You must have created a project on Rational® Test Automation Server and configured a secrets collection in your project.

You must be a member with access to the secrets collection.

1. To grant access to a secrets collection while configuring a new project in Rational® Test Automation Server UI, select the secrets collection listed in the **SECRETS** tab in the Project Configuration page.
2. Alternatively, to grant access to a secrets collection in an existing project, complete the following tasks:
  - a. Log in to Rational® Test Automation Server and from the UI open the project listed under **My Projects**.
  - b. Open the secrets collection from the **SECRETS** tab in the Project Configuration page.  
If there are multiple secrets collections in the project, select the secrets collection that you want from the list.
3. To grant access to a secrets collection in a new project or an existing project, select from the following methods:
  - To add all members with a specific role, click the role listed under **Grant access to role**. For example, if you select **Testers**, then all members in the project with a tester role are granted access to the secrets collection. You can select any role or all the roles listed.
  - To select specific members to grant access to the selected secrets collection, enter the name or the email ID of the member in the field box and add them from the list that is displayed.




**Note:** Members added specifically are listed under **Members with access to this collection** but all the members granted access solely due to their roles are not listed.



**Important:** Irrespective of the role that the member (*Owner, Tester or Viewer*) was assigned in the project, the access to the secrets collections has to be specifically granted to the members from the **SECRETS** tab.

### Removing access to a secrets collection

4. To remove access granted to all members with a specific role or a specific member, select from the following methods:
  - To remove all members with a specific role, click the role listed under **Grant access to role** to clear the selection. For example, if **Testers** is selected and you clear it, then all members in the project with a tester role are removed from the access list to the secrets collection.
  - To remove specific members with access to the secrets collection, select the member and click the **Delete** icon .



**Notes:**





- Any member with access to the secrets collection can remove access of other members specifically added or of all members with a specific role.
- Members with access to the secrets collection can remove themselves from the access list. Members can do this when there is at least one member remaining in the list. After removing themselves, members cannot add themselves back to the access list and must be added by any of the other remaining members in the list.

## Results

You have added members from your project or members with specific role to the access list of people who can access secrets in the selected secrets collection, or you have removed specific members or members with specific role from the access list.

## What to do next

You can create secrets under secrets collections for your project.

## Managing secrets

You can view, edit, or delete the secrets configured under a secrets collection any time after you have created secrets or after you were granted access to the secrets collection. You can change the value of the secret by editing the secret. You can delete secrets that you no longer require in your test environment.

## Before you begin

You must have created a project on Rational® Test Automation Server and configured a secrets collection in your project.


You must have created secrets in the selected secrets collection or the secrets collection must contain secrets.




You must be a member with access to the secrets collection.

1. Log in to Rational® Test Automation Server and from the UI open the project listed under **My Projects**.
2. Complete the following steps:
  - a. Open the secrets collection from the **SECRETS** tab in the Project Configuration page.
  - b. Optionally, select the secrets collection that you want from the list if there are multiple secrets collections in the project.

The secrets configured in the selected secrets collection are displayed.

3. Complete the steps for the task you want to perform as listed in the following table:

Task	Steps
Viewing a secret value	Click the <b>Show</b> icon  for the secret you want to view its value, which most likely is a password for the secret.

Task	Steps
	The value configured for the secret is displayed.
Editing a secret value	<p>Click the <b>Edit</b> icon  for the secret you want to edit, and enter a <code>new value</code> for the secret as its <i>Value</i>. The value can be a password for the secret.</p> <p> <b>Note:</b> You can only change the value of the secret.</p> <p>The value of the selected secret is changed.</p>
Deleting a secret	<p>Click the <b>Delete</b> icon  in the row of the secret you want to delete.</p> <p>After deleting it, the secrets list in the collection is removed from the list.</p>

## Results

- You viewed the password configured of the secret under a secrets collection that you created or were granted access.
- You changed the secret value of the secret under a secrets collection in your project.
- You deleted and removed the secret from the selected secrets collection in your project.

## What to do next

You can use secrets in the tests that require these secrets during test runs.

---

### Related information

[Test run configurations on page 249](#)

[Becoming a project member on page 626](#)

## Managing an encrypted dataset

You can use encrypted datasets to limit access to confidential information such as account number or passwords. You can arrange data by an appropriate category so that project members can use datasets more effectively in certain tests and protect them.

### About this task

A dataset can contain classified information that other members can access with permission. As a project owner, you can group encrypted datasets into different classifications and enable project members to view and edit datasets and run tests associated with the encrypted datasets. After you have created a classification, you can change the classification for a dataset. You can also delete a classification if you do not require it in your test environment.

**Notes:**

- You must grant access and provide an encryption key of an encrypted dataset to other members of the project to work with the encrypted dataset.
- A project member who has been added as a **Tester** role can work with the encrypted dataset.

## Creating a classification

As a project **Owner**, you can organize encrypted datasets by creating a classification so that project members can use and protect datasets more efficiently.


### Before you begin

You must have completed the following tasks:






- Created a project in . See [Test assets and a server project on page 616](#).
- Configured the repository that contains the test assets in your project. See [Adding repositories to a server project on page 620](#).
- Created at least one dataset and encrypted the dataset with an encryption key. See [Dataset encryption on page 158](#).

### About this task

After creating a classification, you can grant access and provide the encryption key to other members of the project to work with the encrypted dataset.

1. Log in to if you are not already logged in.
2. Open a project from the team space.
3. Click **Manage**, and then the **DATA SECURITY** tab.
4. Click **New classification** and enter a name for the classification.
5. Click **Create**.
6. Click the **Add** icon  to select the encrypted dataset to become part of the new classification.
7. Select a dataset from the list and enter the encryption key in the **Password** field for the dataset, and then click the **Add** icon.

The encrypted dataset that is added to the classification is displayed.

DETAILS	REPOSITORIES	SECRETS	DATA SECURITY
Classification			 
Credentials			▼
Datasets			
Name	Encrypted Columns	Actions	
JqueryAccordion	Tag programming languages_ _2		

### What to do next

You can grant access to other project members to use the encrypted dataset.

## Editing or deleting a classification

After you have created a classification, you can edit the name of the classification. You can also delete a classification when it is not required in your test environment.

### Before you begin

You must have completed the following tasks:

- 
- Created at least two classifications. See [Creating a classification on page 647](#).

### About this task

You can opt to edit or delete a classification for your project any time after you create a classification. For example, you might want to edit the name of the classification or delete the classification if the classification that is created earlier are not required.



**Note:** You must be a project **Owner** to create, edit, or delete a classification.


- To edit a classification, go to [Step 1 on page 648](#).
- To delete a classification, go to [Step 2 on page 649](#).

1. Perform the following steps to edit a classification:

- a.
- b.


- c. Select a classification that you want to edit from the list.



- d. Click the **Edit** icon  to edit the selected classification.

- e. Edit the name of the classification, and then click **Save**.

2. Perform the following steps to delete a classification:

- a.
- b. Select a classification that you want to delete from the list.
- c. Click the **Delete** icon .
- d. Click **Delete** in the **Delete classification** window to confirm the deletion of the classification.



**Note:** Before deleting a classification, you must move the added encrypted datasets to another classification else the delete icon is disabled.


## Results

- You have edited the name for a classification in your project.
- You have deleted a classification.

## Moving an encrypted dataset to another classification






When you add many encrypted datasets to the same classification, you can move some of them to another classification.

### Before you begin

- 1.
- 2.
3. Select a classification from the list that has the encrypted dataset.
4. Click the **Edit** icon  from the **Actions** column of a dataset.

5. Select a classification from the list and enter the encryption key of the dataset in the **Password** field.
6. Click **Save**.

A classification for a dataset is updated successfully.

DETAILS	REPOSITORIES	SECRETS	DATA SECURITY
Classification			 
Imp Credentials			▼
Datasets			
Name	Encrypted Columns	Actions	
JqueryAccordion	Tag programming languages_ _2		

## Results


You have moved the encrypted dataset from one classification to another.

## Removing a dataset from the classification

You can remove a datasets added to a classification when they are no longer required.

### Before you begin

You must have completed the following tasks:

- - Created a classification and added the encrypted dataset to it. See [Creating a classification on page 647](#).
- 1.
  - 2.
  3. Select a classification from the list that has an encrypted dataset.
  4. Click the **Delete** icon  from the **Actions** column of a dataset.
  5. Click **Remove** in the **Change the classification for the Dataset** window.



**Note:** Removing dataset from the classification also removes the password stored in for encrypted data. You must enter the password again to gain access to the encrypted columns.

## Results

You have removed the datasets from a classification.

## Granting classification access to members or members roles

You can grant or revoke access to the classification in your project to individual members with different roles or the all members with a specific role. Without access to the classification, members cannot view, create, edit, delete, or use the classification.

### Before you begin

You must be a project **Owner** and have completed the following tasks:

- - Added one or more users to your project. See [Adding users to a project on page 623](#).
  - 
  - Created at least one classification. See [Creating a classification on page 647](#).
- 1.
  - 2.
  3. Select the classification from the drop-down list.
  4. Choose any of the following methods to grant access to a member:

#### Choose from:

- To add all members with a specific role, click the role listed under **Grant access to role**. For example, if you select **Testers** then all members in the project with a tester role are granted access to the selected classification. You can select any role or all the roles listed.
- To select specific members to grant access to the selected classification, enter the name or the email ID of the member in the **Grant access to member** field and add them from the list that displays.




**Note:** Members added specifically are listed under **Members with access to this classification** but the members added for a role are not displayed.



**Important:** Irrespective of the role that the member (**Owner**, **Tester** or **Viewer**) was assigned in the project, the access to the classification has to be specifically granted to the members from the **DATA SECURITY** tab.

5. Choose any of the following methods to revoke access from a member:

#### Choose from:

- To remove all members with a specific role, click the role listed under **Grant access to role** to clear the selection. For example, if **Testers** is selected and you clear it, then all members in the project with a tester role are removed from the access list to the classification.
- To remove specific member with access to the classification, select the member and click the **Delete** icon .

**Notes:**

- Any member with access to the classification can remove access of other members specifically added or of all members with a specific role.
- Members with access to the classification can remove themselves from the access list provided that there is at least one member in the list. After removing themselves, members cannot add themselves back to the access list and must be added by any of the other members in the list.

**Results**

You have granted or revoked the classification access to project members.

---

Related information

[Adding a project on page 619](#)

## Managing notifications

IBM® Rational® Test Automation Server provides a feature to display notifications for different events within the user interface (UI) of Rational® Test Automation Server. You can configure an SMTP server on Rational® Test Automation Server when you want Rational® Test Automation Server to send out notifications about the different events as emails to the subscribed users.

Rational® Test Automation Server provides you the following features to receive notifications about server events:

Channels	Description
In-app	Notifications are displayed within the UI of Rational® Test Automation Server.
SMTP	Notifications are sent in emails to subscribed users of Rational® Test Automation Server by an SMTP server.

### Managing notifications in the server UI

Viewing the in-app notifications or notifications in the server UI is a feature that is enabled automatically when you register as a user in Rational® Test Automation Server.

When you register as a user in Rational® Test Automation Server, the in-app notifications are automatically subscribed for you.

If you want to receive notifications about specific events and not all events, you can change the default subscriptions and modify your subscription for the server events. You can also view the in-app notifications that are displayed for you. See [Managing in-app notifications on page 653](#).



## Managing SMTP notifications

You, as a *Server Administrator* or *Team Space Owner* can configure an existing Simple Mail Transfer Protocol (SMTP) server so that Rational® Test Automation Server can send out notifications about the server events in emails to the registered users who have subscribed to the SMTP notifications. See [Configuring an SMTP server to manage email notifications on page 661](#).

If you, as a registered user of Rational® Test Automation Server want to receive the email notifications about specific events and not all events, you can change the subscriptions for the server events. See [Managing email notifications on page 658](#).

## Managing in-app notifications

You can manage the in-app notifications by changing the subscriptions that are automatically enabled for you for all server events. You can view the server events that are automatically subscribed and then change the subscription for any or all events. For example, as a team space owner, you can choose to be notified when members of the team space create new projects or delete their projects but not be notified when tests are run.


### Before you begin

- 
- 

### About this task

The in-app notifications that are automatically subscribed for you, are displayed in the server UI under the following categories for the server events that occur:


Category	Notifications displayed
User	Include information about actions performed by team space members in the team space.
System	Include information about actions performed by registered users at the server level.

- 1.
2. Select from the following actions that you want to perform:
  - Modify notification settings. Go to [Step 3 on page 653](#).
  - View notifications. Go to [Step 4 on page 655](#).
  - Unsubscribe from the in-app notifications. Go to [Step 5 on page 656](#).
3. Perform the following steps to modify the notification settings:
  - a. Click the **User** icon  on the menu bar.
  - b. Click **Notification Settings**.

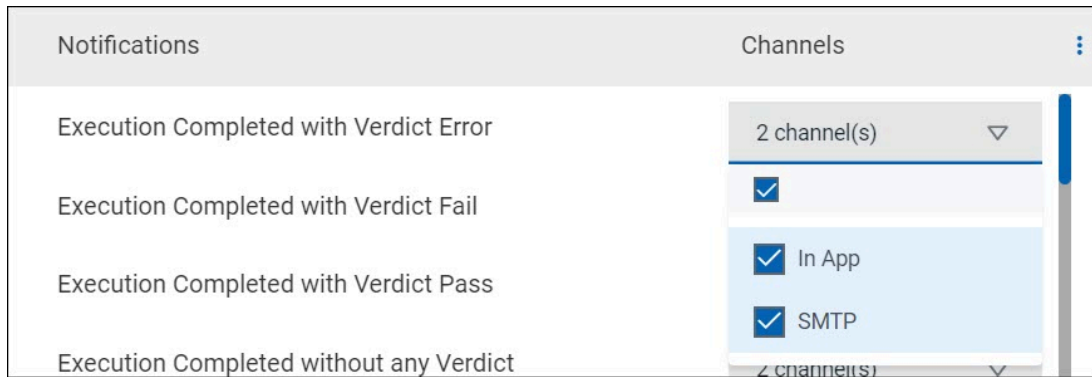
The **Notification Settings** dialog is displayed.

- c. Change the language in which you want to receive the notifications, if you do not want to use the default language by selecting a language from the list.

The default language is the language set on the computer that hosts .

- d. Identify the server events for which you want to change the subscription, and then click the Expand icon  in the row of the server event.

You can notice that for each of the server events, the in-app channel is displayed as selected, which is the default selection unless you have modified the subscription.




- e. Perform the actions indicated in the following table to clear the subscriptions for server events that you do not want to be notified and to select the in-app subscription for the server events that you want to be notified.

When	Action
<p>You do not want to receive the in-app notifications for a specific server event.</p>	<p>Perform the following steps:</p> <ul style="list-style-type: none"> <li>i. Expand the channel list in the row of the server event.</li> <li>ii. Clear the checkbox in the <b>In App</b> row to clear the subscription.</li> <li>iii. Click <b>Apply</b>.</li> </ul> <p>You have unsubscribed from the in-app channel and the in-app notifications for the server event is not displayed for you when the event occurs.</p>
<p>You want to receive the in-app notifications for a specific server event.</p>	<p>Perform the following steps:</p> <ul style="list-style-type: none"> <li>i. Expand the channel list in the row of the server event.</li> <li>ii. Select the checkbox in the <b>In App</b> row, if it is cleared.</li> <li>iii. Click <b>Apply</b>.</li> </ul>

When	Action
	You have subscribed to the in-app channel and the in-app notifications for the server event is displayed for you when the event occurs.

f. Go to [Results on page 658](#).

4. Perform the following steps to view the in-app notifications:

a. Click the **Notifications** icon  on the menu bar.

The **Notifications** dialog is displayed.

The notifications are displayed in the following tabs:


- User
- System

You can find the information about the default notifications that you can view based on your role in the server as in-app notifications in the **USER** tab and **SYSTEM** tab.

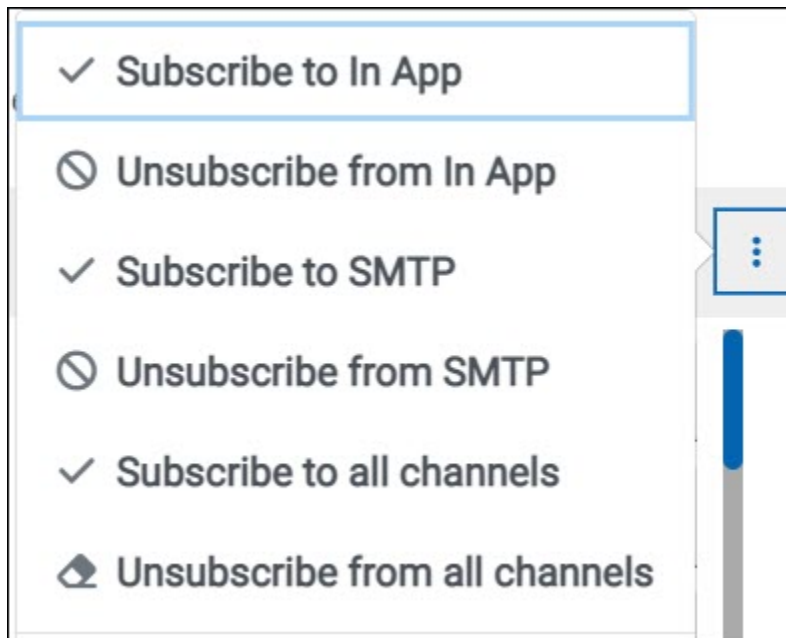
Category	Notification details	or	or		
User notifications	Information about events at the server level or across team spaces performed by other team space owners.	✓	✓	✗	✗
	Information about events in the team space performed by other team space members.	✓	✗	✗	✗
	Information about events in the project performed by other project members.	✗	✓	✓	✗
	Information about events performed by you in a project.	✗	✗	✓	✓
System notifications	Information about events at the server level or across team spaces.	✓	✓	✗	✗

Category	Notification details	or	or		
	Information about events in the team space performed by other team space members.	✓	✓	✗	✗
	Information about events in the projects at the team space level performed by other project members.	✓	✓	✗	✗

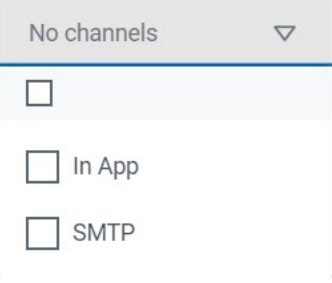
The notifications in the **USER** tab is displayed as the default option.

- b. View the notifications in the **USER** tab.
  - c. View the notifications at the server level by clicking the **SYSTEM** tab.
  - d.
5. Perform the following steps to unsubscribe to the in-app notifications:
- a.
  - b.
  - c. Click the **Open action menu** icon  in the row of **Notifications**.

The following panel is displayed:



d. Perform the actions indicated in the following table for the option you want:

When	Action	Results in
<p>You do not want to receive any in-app notifications.</p>	<p>Click the <b>Unsubscribe from In App</b> option, and then click <b>Apply</b>.</p>	<p>The in-app channel selection for all server events is cleared and you are unsubscribed from the in-app channel. The subscription to the SMTP channel is not changed.</p> <p>If you had subscribed to the SMTP notifications for all events, the SMTP channel is displayed as selected. If you had not subscribed to the SMTP channel, then the SMTP checkbox is not selected.</p>
<p>You do not want to receive any in-app or the SMTP email notifications.</p>	<p>Click the <b>Unsubscribe from all Channels</b> option, and then click <b>Apply</b>.</p>	<p>The in-app channel and the SMTP channel selections for all server events are cleared and you are unsubscribed from both the channels. No channel is displayed as selected for subscription for all server events.</p> 
<p>You want to subscribe to the in-app notifications for all server events.</p>	<p>Click the <b>Subscribe to In App</b> option, and then click <b>Apply</b>.</p>	<p>The in-app channel for all server events is selected and you are subscribed to the in-app channel. The in-app notifications is displayed as selected for subscription for all server events.</p>

When	Action	Results in
You want to receive notifications as in-app and as SMTP email notifications.	Click the <b>Subscribe to all Channels</b> option, and then click <b>Apply</b> .	The in-app channel and the SMTP channel are displayed as selected for all server events and you are subscribed to both the channels. However, the SMTP notifications are sent to your email only if the SMTP server is configured on .

e.

## Results

You completed the following tasks:

- Modified the notification subscriptions.
- Viewed the notifications displayed for you in the **USER** tab and **SYSTEM** tab.
- Unsubscribed from or subscribed to either the in-app channel or both the in-app and SMTP channels.

---

Related information

[Managing notifications on page 652](#)

## Managing email notifications

You can manage the email notifications by changing your subscription to the SMTP channel. For example, as a project owner, you can choose to be notified when members of the project add a repository or run a test in a team space but not be notified when projects are created in the team space.

### Before you begin

You must have completed the following tasks:

- Ensured that the *Server Administrator* or *Team Space Owner* has configured a Simple Mail Transfer Protocol (SMTP) server. See [Configuring an SMTP server to manage email notifications on page 661](#).
- Ensured that you are assigned a role as a *Member* or *Project Creator* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that you are assigned a role as a *Viewer*, *Tester*, or *Project Owner* in the project.

### About this task

After the SMTP server is configured on Rational® Test Automation Server, you can subscribe to the SMTP channel so that you can receive email notifications for server events. You can view the server events that are subscribed in the **Notifications Settings** dialog. You can either unsubscribe from the SMTP channel or change your subscriptions for the server events for which you do not want the email notifications.

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Click the **User** icon  on the menu bar.
3. Click **Notification Settings**.

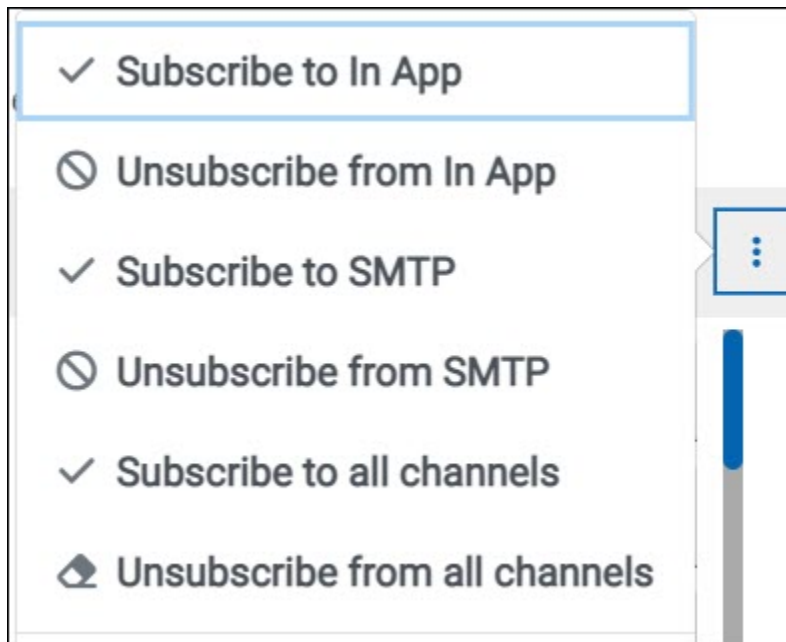
The **Notification Settings** dialog is displayed.

4. Change the language in which you want to receive the email notifications, if you do not want to use the default language by selecting a language from the list.

The default language is the language set on the computer that hosts Rational® Test Automation Server.

5. Click the **Open action menu** icon  in the row of **Notifications**.

The following panel is displayed:



You can notice that the **Subscribe to In App** channel is displayed as selected, unless you modified the default in-app subscription previously.

6. Perform the actions indicated in the following table for the option you want for the email notifications:

When	Action	Results in
You want to subscribe to the email notifications in the SMTP channel for all server events.	Click the <b>Subscribe to SMTP</b> option, and then click <b>Apply</b> .	<p>The SMTP channel for all server events is selected and you are subscribed to the SMTP channel.</p> <p>The <b>SMTP</b> option is displayed as selected for subscription for all server events.</p>
You do not want to receive any email notifications.	Click the <b>Unsubscribe from SMTP</b> option, and then click <b>Apply</b> .	<p>The SMTP channel selection for all server events is cleared and you are unsubscribed from the SMTP channel.</p> <p>The <b>SMTP</b> option is displayed as cleared for subscription for all server events.</p>
You want to receive the email notifications for a specific server event.	Perform the following steps: <ol style="list-style-type: none"> <li>Expand the channel list in the row of the server event.</li> <li>Select the <b>SMTP</b> checkbox.</li> <li>Click <b>Apply</b>.</li> </ol>	You have subscribed to the SMTP channel to receive email notifications for the server event.
You do not want to receive the email notifications for a specific server event, if you subscribed to it previously.	Perform the following steps: <ol style="list-style-type: none"> <li>Expand the channel list in the row of the server event.</li> <li>Clear the checkbox for the <b>SMTP</b> channel option.</li> <li>Click <b>Apply</b>.</li> </ol>	You have unsubscribed from the SMTP channel for the server event and the email notifications are not sent to you when the event occurs.
You want to receive notifications as in-app and as SMTP email notifications.	Click the <b>Subscribe to all Channels</b> option, and then click <b>Apply</b> .	The in-app channel and the SMTP channel are displayed as selected for all server events and you are subscribed to both the channels.



When	Action	Results in
You do not want to receive any in-app or the SMTP email notifications.	Click the <b>Unsubscribe from all Channels</b> option, and then click <b>Apply</b> .	The in-app channel and the SMTP channel selections for all server events are cleared and you are unsubscribed from both the channels. No channel is displayed as selected for subscription for all server events.

## Results

You completed the following tasks:

- Subscribed to the SMTP channel for receiving email notifications for the selected server events.
- Unsubscribed from the SMTP channel for selected server events that you subscribed previously.
- Subscribed to the SMTP channel for receiving email notifications for all server events.
- Unsubscribed from the SMTP channel for receiving email notifications for all server events.
- Subscribed to or unsubscribed from the in-app and SMTP channels.

---

Related information

[Managing notifications on page 652](#)

## Configuring an SMTP server to manage email notifications

Before you can receive email notifications about events that occur in Rational® Test Automation Server, you need to configure an existing Simple Mail Transfer Protocol (SMTP) server in Rational® Test Automation Server. When events occur in Rational® Test Automation Server, email notifications are sent to the registered users of Rational® Test Automation Server if they are subscribed to the SMTP notification channel.

### Before you begin

You must have completed the following tasks:

- Ensured that you are assigned a role as a *Team Space Owner* in the team space. See [Managing members and their roles in a team space on page 612](#).
- Ensured that an existing SMTP server is available to be connected to Rational® Test Automation Server.

### About this task

The configuration works for emails that are supported by the SMTP server. To use this capability, you must enable IBM® Rational® Test Automation Server to connect to the SMTP server to send email notifications.

As a server administrator, you can configure email notifications so that other users can receive information about actions on the server projects.

1. Log in to Rational® Test Automation Server.

The team space that contains your project is displayed.

2. Click the **Settings** icon  from the menu bar.

The **Notifications** page is displayed.

3. Click **Add Configuration**.
4. Provide the following details to configure the SMTP server:
  - A name to the configuration.
  - The host name and port number of the SMTP server.



**Note:** The default configuration uses a local SMTP server that listens to the default port 25. This SMTP server does not require any authentication.

- The email address of the sender. The email address is displayed in email notifications. The email notifications are sent to the users based on the notification settings.
- The encryption type for secure communications is when the SMTP server is on an external network. You can enable either **STARTTLS** or **SSL/TLS**.



**Note:** The SMTP server for the secure communication listens to the default port 465.

- Login credentials are required if the SMTP server requires authentication.

5. Click **Add**.



**Note:** The SMTP configuration is added after a valid connection is established with the SMTP server.

## Results

You configured an SMTP server to connect with Rational® Test Automation Server and send out email notifications about the server events to users who have subscribed to the SMTP channel.

## What to do next

You can modify the default subscriptions for the SMTP channel, see [Managing email notifications on page 658](#).

---

### Related information

[Managing notifications on page 652](#)

# Chapter 8. Troubleshooting

This guide describes how to analyze and resolve some of the common problems that you might encounter while you work with Rational® Test Automation Server.

In addition to the list of problems that you might encounter, these resources are available to help you troubleshoot Rational® Test Automation Server problems. Known problems of Rational® Test Automation Server are documented in the form of individual technotes in the Support Knowledge Base. As problems are discovered and resolved, the knowledge base is updated and maintained with new information. By searching the knowledge base, you can quickly find workarounds or solutions to problems.

[Troubleshooting documentation for Rational® Test Workbench](#)

You can contact IBM® support if you are unable to troubleshoot the problem. Gather all the required background information and provide the details to IBM® support for investigation. For more information, see [IBM Support](#).

## Troubleshooting issues

You can find information about the issues or problems that you might encounter while working with Rational® Test Automation Server. Details about issues, their causes and the resolutions that you can apply to fix the issues are described.

The troubleshooting issues are presented to you in the following tables based on where or when you might encounter these issues on Rational® Test Automation Server.

- [Table 17: Troubleshooting issues: installation on page 663](#)
- [Table 18: Troubleshooting issues: server administration on page 666](#)
- [Table 19: Troubleshooting issues: resource monitoring on page 666](#)
- [Table 20: Troubleshooting issues: configuring test runs on page 667](#)
- [Table 21: Troubleshooting issues: test or stub runs on page 668](#)
- [Table 23: Troubleshooting issues: test results and reports on page 672](#)


**Table 17. Troubleshooting issues: installation**

<b>Problem</b>	<b>Cause</b>	<b>Solution</b>
On Ubuntu, when you are installing the server software and you encounter errors in the scripts that are running.	At times, scripts might not appear to be running due to any of the following reasons:	You can complete any of the following tasks:

Table 17. Troubleshooting issues: installation (continued)

Problem	Cause	Solution
	<ul style="list-style-type: none"> <li>• Slow connection speeds.</li> <li>• Insufficient CPU, memory, or disk resources.</li> <li>• A firewall that was configured incorrectly is already enabled.</li> </ul>	<ul style="list-style-type: none"> <li>• To identify the issue, you can perform a diagnostic check by running the following command:           <pre>journalctl -u k3s</pre>           This command displays the log that you can use to check for the problem.         </li> <li>• Run the following command to see which pods are running and which pods are not running:           <pre>kubectl get pods -A</pre>           Run the following command to get details about a specific pod:           <pre>kubectl describe pod -n &lt;namespace&gt; &lt;pod name&gt;</pre> </li> <li>• Follow the on-screen instructions to resolve the errors.</li> <li>• Some issues can be solved by re-running the following script:           <pre>sudo ./ubuntu-init.sh</pre> </li> </ul>
On Ubuntu, DNS is not working as expected.		<p>The DNS configuration that is used by the cluster can be displayed by using the following command:</p> <pre>kubectl get cm -n kube-system coredns -ojsonpath="{.data.Corefile}"</pre>

Table 17. Troubleshooting issues: installation (continued)

Problem	Cause	Solution
		<p>The <b>forward</b> setting displays the <i>nameservers</i> that are used. For example, you might see the following in the <code>corefile</code>:</p> <pre data-bbox="1036 520 1419 716">.:53 { :     forward . 8.8.8.8 9.9.9.9 : }</pre> <p>A script (<code>ubuntu-set-dns.sh</code>) is supplied for managing these values.</p> <p>For example, to set the DNS values for the values shown in the previous example:</p> <pre data-bbox="1036 982 1419 1079">sudo ./ubuntu-set-dns.sh --server 8.8.8.8 --server 9.9.9.9</pre> <p> <b>Note:</b> If you do not use <code>sudo</code> in the command, the script runs but the configuration might be lost if the cluster is restarted.</p> <p>To learn more about the behavior of the script, run the following command:</p> <pre data-bbox="1036 1507 1419 1570">sudo ./ubuntu-set-dns.sh --help</pre>
When running <code>helm install</code> the created pods keep crashing, and the logs contain: <code>ACCESS_REFUSED</code> when trying to connect to RabbitMQ	In some instances, the RabbitMQ password is not automatically setup correctly.	Manually apply the necessary password:

**Table 17. Troubleshooting issues: installation (continued)**

Problem	Cause	Solution
		<pre>&lt;release-name&gt;-rabbitmq -o   jsonpath='{.data.rabbitmq-p   assword}'   base64   --decode)"</pre>

**Table 18. Troubleshooting issues: server administration**

Problem	Cause	Solution
When a user is assigned an additional role, the change in the permissions is not observed in the browser.		You must log out of the session and log in again for the changed role to take effect.
You see the following message displayed on Rational® Test Automation Server:  <div style="border: 1px solid black; padding: 5px; width: fit-content;">       You can't request to join a project that has no owners     </div>	You requested to join a project that no longer has an owner. Orphaned projects occur when the project owners are deleted. This can occur, for example, when the person leaves the organization.	Ask an administrator to take ownership of the project, and then add you as a member.

**Table 19. Troubleshooting issues: resource monitoring**

Problem	Cause	Solution
You are not able to add a Prometheus server as a Resource Monitoring source.	The cause might be that you have not installed the Prometheus server at the time of server installation.	Verify that the Prometheus server was installed in Helm at the time of server installation. See <a href="#">Installing the server software on Ubuntu by using k3s on page 53</a> . If not, consult your cluster administrator to get the Prometheus server installed and configured.

Table 20. Troubleshooting issues: configuring test runs

Problem	Cause	Solution
<p>When you configure a run of a schedule that matches the following conditions:</p> <ul style="list-style-type: none"> <li>• The schedule has two user groups configured to run on static agents when the schedule was created in Rational® Performance Tester 10.1.</li> <li>• One of the user groups is disabled and the asset is committed to the remote repository.</li> </ul> <p>Both the static agents are displayed as available for the test run in the <b>Location</b> tab of the <b>Execute test asset</b> dialog when only one agent that is configured for the user group must be available.</p>	<p>The cause might be because of the following reasons:</p> <ul style="list-style-type: none"> <li>• The schedule was created in Rational® Performance Tester 10.1.</li> <li>• The user group that is disabled is not removed or deleted from the test resources.</li> <li>• The agent configured on the disabled user group is already added as an agent to the server project and is available for selection.</li> </ul>	<p>To resolve the problem, select from either of the following methods:</p> <ul style="list-style-type: none"> <li>• By using Rational® Performance Tester 10.1.1.</li> </ul> <p>Perform the following steps:</p> <ol style="list-style-type: none"> <li>1. Open the schedule in Rational® Performance Tester 10.1.1.</li> <li>2. Save the schedule and the project.</li> <li>3. Commit your test asset to the remote repository.</li> <li>4. Proceed to configure a run for the schedule on Rational® Test Automation Server 10.1.1.</li> </ol> <ul style="list-style-type: none"> <li>• By using Rational® Performance Tester 10.1.</li> </ul> <p>Perform the following steps:</p> <ol style="list-style-type: none"> <li>1. Select the disabled user group.</li> <li>2. Click <b>Remove</b>.</li> <li>3. Save the schedule and the project.</li> <li>4. Commit your test asset to the remote repository.</li> <li>5. Proceed to configure a run for the schedule on Rational® Test Automation Server 10.1.1.</li> </ol>

**Table 20. Troubleshooting issues: configuring test runs (continued)**

Problem	Cause	Solution
<p>You have added a remote repository to your project that contains the test assets or resources of the following types:</p> <ul style="list-style-type: none"> <li>• Postman</li> <li>• JMeter</li> <li>• JUnit</li> </ul> <p>The test assets or resources are not displayed on the <b>Execution</b> page for you to select the asset for a run.</p>	<p>This problem occurs if the server extension is not enabled. Although the extension was enabled at the time of installation of Rational® Test Automation Server, it was disabled subsequently by the server administrator.</p>	<p>You must verify if the server extension is enabled and running by running the following command: <code>kubectl get pod -n &lt;test-system&gt;</code>, where <code>&lt;test-system&gt;</code> is the namespace that you created to install the server software. The server extensions that are running are displayed.</p> <p>If the server extension that you want is not running implying that the server extension is not enabled. You must enable the server extension. Contact the server administrator to enable the server extension.</p>

**Table 21. Troubleshooting issues: test or stub runs**



Problem	Cause	Solution
<p>You encounter any of the following issues:</p> <ul style="list-style-type: none"> <li>• When many tests are run simultaneously on the default cluster location and you observe the following issues: <ul style="list-style-type: none"> <li>◦ Out of memory errors.</li> <li>◦ Observe that the test runs are slow with a high CPU usage.</li> <li>◦ The Kubernetes pods are getting evicted.</li> </ul> </li> <li>• When you run an AFT suite that contains multiple Web UI tests and you observe the following issues:</li> </ul>	<p>The issue is seen when any of the following events occur:</p> <ul style="list-style-type: none"> <li>• Many tests are run in parallel.</li> <li>• The memory that is used by the tests during the test run exceeds the allocated default memory of 1 GB.</li> <li>• The default memory of the container is not adequate for the test run.</li> <li>• Pods are evicted due to low node memory.</li> </ul>	<p>To resolve the problem, you can increase the resource allocation for test runs by using the arguments listed in <a href="#">Table 22: Increasing resource allocation on page 670</a>.</p> <p>You can enter arguments in the <b>Additional configuration options</b> field in the <b>Advanced</b> settings panel of the <b>Execute test asset</b> dialog when configuring a test run.</p> <p> <b>Important:</b> The memory settings that you configure for a test run is persisted for the test when ever you run it. You must use this setting judiciously. Configuring all</p>



Table 21. Troubleshooting issues: test or stub runs (continued)

Problem	Cause	Solution
<ul style="list-style-type: none"> <li>◦ Error stating that the browser might not be installed or the browser version is unsupported.</li> <li>◦ Error stating multiple random time-outs or an internal error.</li> <li>• When you run VU schedules that contain multiple Web UI test or large number of virtual users or a combination of Web UI, performance, and API tests, and you observe the following issue: <ul style="list-style-type: none"> <li>◦ The test run hangs and the inadequacy of memory can be inferred as the cause in the logs.</li> </ul> </li> </ul>		<p> tests for an increased memory limit might affect subsequent test runs or cause other memory issues when tests run simultaneously.</p> <p>In addition, in the <b>JVM Arguments</b> field under the <b>Advanced</b> settings you can set the maximum heap size for the test run time. For example, adding the JVM argument <b>-Xmx3g</b> sets the maximum heap size to 3Gi.</p>
<p>You are not able to run the Istio stubs from the <b>Execution</b> page.</p>	<p>The cause might be that the fully qualified domain name is not specified in the <b>Host</b> field for the stub when it was created.</p>	<p>Verify and ensure to add the fully qualified domain name of the server in the <b>Host</b> field when the physical transport for the stub is configured in Rational® Integration Tester.</p>
<p>You are not able to run Web UI tests on a remote agent that runs on Linux because the browser fails to launch.</p>	<p>Browser launch is prevented when the remote agent runs as <i>root</i> or <i>sudo</i>.</p>	<p>To resolve this issue, you must perform the following steps after you have installed the remote agent:</p> <ol style="list-style-type: none"> <li>1. Stop the agent.</li> <li>2. Start the agent as the logged in user by performing the following steps: <ol style="list-style-type: none"> <li>a. Open a terminal on Linux.</li> <li>b. Enter the following commands:</li> </ol> </li> </ol>

**Table 21. Troubleshooting issues: test or stub runs (continued)**

Problem	Cause	Solution
		<pre>cd /opt/IBM/SDP/Ma- jorjomo/  sudo ./MDStop.sh  ./MDStart.sh</pre>




**Note:** You can refer to the [Kubernetes documentation](#) for information about the different units that can be used for resources in the **Additional configuration option** fields.

**Table 22. Increasing resource allocation**

Requirement	Configuration option name	Default value, if no value is set	An example value	Result of using the example value
Specifying the memory limit of the <i>init container</i> .	<code>init.resource.memory.limit</code>	<i>1024Mi</i>	<i>2048Mi</i>	Increases the memory limit of the <i>init container</i> from the default value to <i>2048Mi</i> .
Configuring a larger memory request for the <i>init container</i> to avoid pod eviction.	<code>init.resource.memory.request</code>	<i>64Mi</i>	<i>1024Mi</i>	Increases the initial memory request for the <i>init container</i> from the default value to <i>1024Mi</i> .
Specifying the cpu request for the <i>init container</i> .	<code>init.resource.cpu.request</code>	<i>50m</i>	<i>60m</i>	Increases the cpu request for the <i>init container</i> from the default value <i>60m</i> .
Specifying the memory limit of the container used for the test run.	<code>resource.memory.limit</code>	The larger of <i>3Gi</i> or <i>maximum heap size + 1Gi</i>	<i>4Gi</i>	Changes the memory limit of the main con-

Table 22. Increasing resource allocation

(continued)

Requirement	Configuration option name	Default value, if no value is set	An example value	Result of using the example value
<p> <b>Note:</b> If the memory limit you set is more than the default limit, you must run the following command to increase the allotted limit to 8 GB or any other value above the default value of 3 GB, at the time of installing the server or any time later:</p> <pre data-bbox="285 1234 418 1409">--set   execution   .template   resources   limits.memory=8Gi</pre> <p>You must be a server administrator to run this command.</p>				tainer from the default value to <i>4Gi</i> .
Specifying the memory request for the container used by the test run.	<code>resource.memory.request</code>	<i>1024Mi</i>	<i>2048Mi</i>	Increases the memory request for the main container from

**Table 22. Increasing resource allocation**

(continued)

Requirement	Configuration option name	Default value, if no value is set	An example value	Result of using the example value
				the default value to <i>2048Mi</i> .
Specifying the cpu request for the main container used by the test run.	<code>resource.cpu.request</code>	<i>50m</i>	<i>70m</i>	Increases the cpu request for the main container from the default value to <i>70m</i> .

**Table 23. Troubleshooting issues: test results and reports**

Problem	Cause	Solution
You are not able to view the Jaeger traces for the tests you ran.	<p>The cause can be as follows:</p> <ul style="list-style-type: none"> <li>• Jaeger was not pre-installed in Red Hat OpenShift.</li> <li>• The Jaeger trace is not supported for the particular test that you ran.</li> </ul>	<p>Check for any of the following solutions:</p> <ul style="list-style-type: none"> <li>• Verify that Jaeger was installed in Helm at the time of server installation. See <a href="#">Installing the server software on Ubuntu by using k3s on page 53</a>. If not, consult your administrator to get Jaeger installed and configured.</li> <li>• Verify that the tests you ran are supported for Jaeger traces. See <a href="#">Test results and reports on page 456</a>.</li> <li>• Verify if you provided the program variables when you configured the test run. See <a href="#">Considerations for using Jaeger traces in reports on page 205</a>.</li> </ul>

# Security Considerations

This document describes the actions that you can take to ensure that your installation is secure, customize your security settings, and set up user access controls.

- [Enabling secure communication between multiple applications on page dclxxiii](#)
- [Ports, protocols, and services on page dclxxiii](#)
- [Customizing your security settings on page dclxxiii](#)
- [Setting up user roles and access on page dclxxiv](#)

## Enabling secure communication between multiple applications

The majority of communications are sent over TLS to port 443 (see [Ports, protocols, and services on page dclxxiii](#)). During the installation, an X.509 certificate is generated for the user provided DNS name, which is used to connect to the server. This certificate is self-signed and hence untrusted by other applications.

This self-signed certificate must be replaced by a certificate signed by a certificate authority trusted by your organization. For more information, see [X.509 Certificate User Authentication](#) in the Keycloak documentation.

For information about how the self-signed certificate was created, see the `ssl.sh` file in the `<install-directory>/prepare/` directory.

For information about importing a certificate authority trusted by your organization, see [Certificate authority: Importing and extending lists on page 115](#).

## Ports, protocols, and services

TCP port 443 is used by the majority of communications with the server.

The port 7085 is the default port for communications with agents registered with Rational® Test Automation Server.

The ports starting from 7085, are used in pairs such as 7085 and 7086, and are allotted for the Schedule that is executed first. The next Schedule is allotted the next pair (7087,7088), and so on for the Schedules that are running simultaneously.

You must open the required ports in pairs for each of the Schedules that you want to run simultaneously.

## Customizing your security settings

You can customize your security settings through user registration.

### User registration

By default, users can sign up themselves with the server. In some environments, this self sign-up might be undesirable. It can be changed by switching off user registration. For more information, see [User Registration](#) in the Keycloak documentation.

By default, user email addresses are not verified. This verification must be enabled in production environments. For more information, see [Email settings on page 109](#).

## **Setting up user roles and access**

You can manage user roles and access through single sign on (SSO) and administration only accounts.

### **Single sign-on**

By default, Keycloak manages users and passwords locally. In production environments, it is normally appropriate to use single sign-on. For more information, see [LDAP user administration on page 109](#).

### **Administration only accounts**

Users in the Administrator group can manage the team space where they can configure licenses and notifications and add a repository to store the System model. For more information, see [Team space overview on page 126](#).

Users in the Administrator group can also discover all projects stored on the server (including private ones) and assign themselves and others roles in those projects.

For this reason, users who use the server to perform both administration and non-administration tasks must have two different accounts, one for each purpose. For more information, see [Default user administration on page 107](#).

# Notices

This document provides information about copyright, trademarks, terms and conditions for the product documentation.

© Copyright IBM Corporation 2000, 2016 / © Copyright HCL Technologies Limited 2016, 2022

This information was developed for products and services offered in the US.

IBM® may not offer the products, services, or features discussed in this document in other countries. Consult your local IBM® representative for information on the products and services currently available in your area. Any reference to an IBM® product, program, or service is not intended to state or imply that only that IBM® product, program, or service may be used. Any functionally equivalent product, program, or service that does not infringe any IBM® intellectual property right may be used instead. However, it is the user's responsibility to evaluate and verify the operation of any non-IBM® product, program, or service.

IBM® may have patents or pending patent applications covering subject matter described in this document. The furnishing of this document does not grant you any license to these patents. You can send license inquiries, in writing, to:

*IBM Director of Licensing*  
*IBM Corporation*  
*North Castle Drive, MD-NC119*  
*Armonk, NY 10504-1785*  
*US*

For license inquiries regarding double-byte character set (DBCS) information, contact the IBM® Intellectual Property Department in your country or send inquiries, in writing, to:

*Intellectual Property Licensing*  
*Legal and Intellectual Property Law*  
*IBM Japan Ltd.*  
*19-21, Nihonbashi-Hakozakicho, Chuo-ku*  
*Tokyo 103-8510, Japan*

INTERNATIONAL BUSINESS MACHINES CORPORATION PROVIDES THIS PUBLICATION "AS IS" WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF NON-INFRINGEMENT, MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. Some jurisdictions do not allow disclaimer of express or implied warranties in certain transactions, therefore, this statement may not apply to you.

This information could include technical inaccuracies or typographical errors. Changes are periodically made to the information herein; these changes will be incorporated in new editions of the publication. IBM® may make improvements and/or changes in the product(s) and/or the program(s) described in this publication at any time without notice.

Any references in this information to non-IBM® websites are provided for convenience only and do not in any manner serve as an endorsement of those websites. The materials at those websites are not part of the materials for this IBM® product and use of those websites is at your own risk.

IBM® may use or distribute any of the information you provide in any way it believes appropriate without incurring any obligation to you.

Licensees of this program who wish to have information about it for the purpose of enabling: (i) the exchange of information between independently created programs and other programs (including this one) and (ii) the mutual use of the information which has been exchanged, should contact:

*IBM Director of Licensing*  
*IBM Corporation*  
*North Castle Drive, MD-NC119*  
*Armonk, NY 10504-1785*  
*US*

Such information may be available, subject to appropriate terms and conditions, including in some cases, payment of a fee.

The licensed program described in this document and all licensed material available for it are provided by IBM® under terms of the IBM® Customer Agreement, IBM® International Program License Agreement or any equivalent agreement between us.

The performance data discussed herein is presented as derived under specific operating conditions. Actual results may vary.

Information concerning non-IBM® products was obtained from the suppliers of those products, their published announcements or other publicly available sources. IBM® has not tested those products and cannot confirm the accuracy of performance, compatibility or any other claims related to non-IBM® products. Questions on the capabilities of non-IBM® products should be addressed to the suppliers of those products.

Statements regarding the future direction or intent of IBM® are subject to change or withdrawal without notice, and represent goals and objectives only.

This information contains examples of data and reports used in daily business operations. To illustrate them as completely as possible, the examples include the names of individuals, companies, brands, and products. All of these names are fictitious and any similarity to actual people or business enterprises is entirely coincidental.

#### COPYRIGHT LICENSE:

This information contains sample application programs in source language, which illustrate programming techniques on various operating platforms. You may copy, modify, and distribute these sample programs in any form without payment to IBM®, for the purposes of developing, using, marketing or distributing application programs conforming to the application programming interface for the operating platform for which the sample programs are written. These examples have not been thoroughly tested under all conditions. IBM®, therefore, cannot guarantee or imply reliability,



serviceability, or function of these programs. The sample programs are provided "AS IS", without warranty of any kind. IBM® shall not be liable for any damages arising out of your use of the sample programs.

Each copy or any portion of these sample programs or any derivative work must include a copyright notice as follows:

© (your company name) (year).

Portions of this code are derived from IBM Corp. Sample Programs.

© Copyright IBM Corp. 2000, 2022.

## Trademarks

IBM®, the IBM® logo, and [ibm.com](http://ibm.com)® are trademarks or registered trademarks of International Business Machines Corp., registered in many jurisdictions worldwide. Other product and service names might be trademarks of IBM® or other companies. A current list of IBM® trademarks is available on the web at "Copyright and trademark information" at [www.ibm.com/legal/copytrade.shtml](http://www.ibm.com/legal/copytrade.shtml).

## Terms and conditions for product documentation

Permissions for the use of these publications are granted subject to the following terms and conditions.

### Applicability

These terms and conditions are in addition to any terms of use for the IBM® website.

### Personal use

You may reproduce these publications for your personal, noncommercial use provided that all proprietary notices are preserved. You may not distribute, display or make derivative work of these publications, or any portion thereof, without the express consent of IBM®.

### Commercial use

You may reproduce, distribute and display these publications solely within your enterprise provided that all proprietary notices are preserved. You may not make derivative works of these publications, or reproduce, distribute or display these publications or any portion thereof outside your enterprise, without the express consent of IBM®.

### Rights

Except as expressly granted in this permission, no other permissions, licenses or rights are granted, either express or implied, to the publications or any information, data, software or other intellectual property contained therein.

IBM® reserves the right to withdraw the permissions granted herein whenever, in its discretion, the use of the publications is detrimental to its interest or, as determined by IBM®, the above instructions are not being properly followed.

You may not download, export or re-export this information except in full compliance with all applicable laws and regulations, including all United States export laws and regulations.

IBM® MAKES NO GUARANTEE ABOUT THE CONTENT OF THESE PUBLICATIONS. THE PUBLICATIONS ARE PROVIDED "AS-IS" AND WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING BUT NOT LIMITED TO IMPLIED WARRANTIES OF MERCHANTABILITY, NON-INFRINGEMENT, AND FITNESS FOR A PARTICULAR PURPOSE.

## IBM Online Privacy Statement

IBM Software products, including software as a service solutions, (Software Offerings) may use cookies or other technologies to collect product usage information, to help improve the end user experience, to tailor interactions with the end user, or for other purposes. In many cases no personally identifiable information is collected by the Software Offerings. Some of our Software Offerings can help enable you to collect personally identifiable information. If this Software Offering uses cookies to collect personally identifiable information, specific information about this offerings use of cookies is set forth below.

This Software Offering does not use cookies or other technologies to collect personally identifiable information.

If the configurations deployed for this Software Offering provide you as customer the ability to collect personally identifiable information from end users via cookies and other technologies, you should seek your own legal advice about any laws applicable to such data collection, including any requirements for notice and consent.

For more information about the use of various technologies, including cookies, for these purposes, see IBM Privacy Policy at <http://www.ibm.com/privacy> and IBM Online Privacy Statement at <http://www.ibm.com/privacy/details> in the section entitled Cookies, Web Beacons and Other Technologies, and the IBM Software Products and Software-as-a-Service Privacy Statement at <http://www.ibm.com/software/info/product-privacy>.

# Index

## A

- accessibility
  - IBM
  - Rational
  - Test Automation Server
    - disability 28
    - keyboard 28
- Accessing secrets 640
- administration default security
  - IBM
  - Rational
  - Test Automation Server 107
- administration ldap security
  - IBM
  - Rational
  - Test Automation Server 109

## B

- backing up and restoring user data
  - IBM
  - Rational
  - Test Automation Server 89, 91, 93

## C

- Change
  - Classification 646
  - Dataset 646
  - Encrypt 646
- Change Classification
  - Dataset 163
- Configure
  - Dataset 163
- Create
  - Classification 646
- Create Dataset 147
- Creating secrets 640

## D

- Dataset 163
- datasets
  - encrypting 158
  - removing encryption 160
- Delete
  - Classification 646
- Deleting secrets 640

## E

- Edit
  - Classification 646
- Editing secrets 640

## G

- getting started
  - guide 19
- Grant
  - access 646
  - Classification 646
- guide
  - getting started 19

## I

- IBM
- Rational
- Test Automation Server
  - accessibility 28

- backing up and restoring user data 89, 91, 93
- default security administration 107
- disability 28
- keyboard 28
- ldap security administration 109
- managing account settings 114
- repository considerations 617
- test assets 616

## IBM

- Rational
- Test Automation Server
  - project overview
  - test assets statistics 630

## M

- Manage encrypted dataset 646
- managing account settings
  - IBM
  - Rational
  - Test Automation Server 114
- managing project secrets
  - Rational
  - Test Automation Server 640

## O

- overview
  - server 19

## R

- Rational
- Test Automation Server
  - managing secrets 640
- Remove
  - Classification 646
  - Dataset 646
  - Encrypt 646
- repository considerations
  - IBM
  - Rational
  - Test Automation Server 617
- Revoke
  - access 646
  - Classification 646

## S

- Secrets
  - accessing 640
  - creating 640
  - deleting 640
  - editing 640
  - viewing 640
- security considerations
  - overview dclxxiii
- server
  - overview 19
  - task flows 24
- Set
  - Current row 163
  - Dataset 163

## T

- task flows
  - server 24
- test assets
  - IBM

- Rational
- Test Automation Server 616

- test assets statistics
  - IBM
  - Rational
  - Test Automation Server
    - project overview 630
- tests
  - datasets 158, 160

## V

- View
  - Dataset 163, 163
  - Encrypt 163
  - Viewing secrets 640