



Toon Boom Harmony 17 Installation Guide



TOON BOOM ANIMATION INC.
4200 Saint-Laurent, Suite 1020
Montreal, Quebec, Canada
H2W 2R2

+1 514 278 8666
contact@toonboom.com
toonboom.com

Legal Notices

Toon Boom Animation Inc.
4200 Saint-Laurent, Suite 1020
Montreal, Quebec, Canada
H2W 2R2

Tel: +1 514 278 8666

Fax: +1 514 278 2666

toonboom.com

Disclaimer

The content of this document is the property of Toon Boom Animation Inc. and is copyrighted. Any reproduction in whole or in part is strictly prohibited.

The content of this document is covered by a specific limited warranty and exclusions and limit of liability under the applicable License Agreement as supplemented by the special terms and conditions for Adobe®Flash® File Format (SWF). For details, refer to the License Agreement and to those special terms and conditions.

Trademarks

Toon Boom® is a registered trademark. Harmony™ and the Toon Boom logo are trademarks of Toon Boom Animation Inc. All other trademarks of the property of their respective owners.

Publication Date

10-27-2020

Copyright © 2020 Toon Boom Animation Inc., a Corus Entertainment Inc. company. All rights reserved.

Table of Contents

Table of Contents	2
Introduction	10
Chapter 1: Basic Installation	12
About Windows Installation	13
Installing Harmony Dependencies on Windows	14
Installing Harmony on Windows	16
Adding Harmony Executables to the %PATH% Environment Variable on Windows	18
About macOS Installation	19
Installing Harmony on macOS	20
Launching Harmony and Side Applications on macOS	21
Adding Harmony Binaries to the \$PATH Environment Variable on macOS	22
About GNU/Linux Installation	23
About GNU/Linux Installation	24
Installing Harmony Dependencies on GNU/Linux	25
Installing Proprietary Video Drivers on GNU/Linux	26
Installing Harmony on GNU/Linux	30
Adding Harmony Binaries to the \$PATH Environment Variable on GNU/Linux	38
About Audio Transcoding on GNU/Linux	39
Preventing the Alt Key Keyboard Shortcut Conflict Between Harmony and the GNU/Linux Desktop ..	40
Troubleshooting Problems with Harmony on GNU/Linux	41
Troubleshooting a Crash When Loading Harmony on GNU/Linux	42
Troubleshooting Desktop Shortcuts Launching the Wrong Version of Harmony	44
Chapter 2: Harmony Server	46
About the Harmony Server Structure	47
About Database Server Installation	49
Database Server Installation on Windows	50
Disabling the Firewall for a Harmony Database Server on Windows	51
Setting up the Database Server on Windows	52
Removing the Scene Data Storage from the Database Server on Windows	54

Running the Database Server Service on Log-In on Windows	55
Opening the Firewall Ports for a Harmony Database Server on Windows	57
Creating a Scene Data Storage Space on Windows	59
Configuring a Windows Database Server to have a Static IP Address	65
Looking Up the Database Server Log on Windows	69
Launching the Database Server Manually on Windows	70
How to Stop and Restart the Database Server on Windows	71
Pausing and Resuming the Database Server on Windows	73
Uninstalling the Database Server on Windows	74
Database Server Installation on macOS	76
Setting the Server Hostname on macOS	77
Creating a Harmony Database on macOS	78
Installing the Database Server Daemon on macOS	79
Running the Database Server on Log-In on macOS	81
Sharing the Harmony Database Files Using NFS on macOS	83
About Supporting Windows Clients for macOS Servers	85
Installing the Link Server Daemon on macOS	86
Running the Link Server on Log-In on macOS	87
Sharing the Harmony Database Files Using Samba on macOS	89
Configuring a macOS Database Server for Windows Clients	93
Creating a Scene Data Storage Server on macOS	95
Configuring a macOS Database Server to have a Static IP Address	99
Looking Up the Database Server Log on macOS	102
Launching the Database Server Manually in macOS	104
Stopping and Restarting the Database Server on macOS	105
Pausing and Resuming the Database Server on macOS	107
Uninstalling the Database Server on macOS	108
Database Server Installation on GNU/Linux	110
Disabling Security for a Harmony Database Server on GNU/Linux	111
Creating a Harmony Database on GNU/Linux	113

Installing the Database Server Daemon on GNU/Linux	114
Running the Database Server on Log-In on GNU/Linux	116
Expanding the File Descriptor Limit for a Database Server on GNU/Linux	118
Opening the Firewall Ports for a Harmony Database Server on GNU/Linux	119
Sharing the Harmony Database Files Using NFS on GNU/Linux	120
About Supporting Windows Clients for GNU/Linux Servers	123
Installing the Link Server Daemon on GNU/Linux	124
Running the Link Server on Log-In on GNU/Linux	126
Opening the Firewall Ports for the Link Server on GNU/Linux	128
Sharing the Harmony Database Files Using Samba on GNU/Linux	129
Opening the Firewall Ports and Configuring SELinux for a Samba Server on GNU/Linux	131
Configuring a GNU/Linux Database Server for Windows Clients	133
Creating a Scene Data Storage Server on GNU/Linux	135
Setting Up Static IPs for WebCC on GNU/Linux	138
Checking the Database Server Log on GNU/Linux	143
Launching the Database Server Manually in GNU/Linux	144
Stopping and Restarting the Database Server on GNU/Linux	145
Pausing and Resuming the Database Server on GNU/Linux	147
Uninstalling the Database Server on GNU/Linux	148
Connecting Client Workstations to a Harmony Database Server	150
Configuring Harmony Server Clients on Windows	151
Connecting a Windows Client to a Database Server Using Configuration Wizard	152
Manually Configuring a Windows Client to Connect to a Harmony Database	153
Configuring the Hosts File for a Windows Client	156
About Connecting to a Harmony Database Without Using Shortcut Files on Windows	157
Configuring the Shortcuts File on Windows	158
Setting a Custom Location for the Shortcuts Configuration File on Windows	160
Troubleshooting Client Connections on Windows	162
Configuring Harmony Server Clients on macOS	166
Connecting a macOS Client to a Database Server	167

Setting the User File Creation Permission Mask on macOS	169
Configuring the Hosts File for a macOS Client	170
About the Location of Mount Points on macOS	171
Configuring the Shortcuts File on macOS	172
Setting a Custom Location for the Shortcuts Configuration File on macOS	173
Configuring Harmony Server Clients on GNU/Linux	174
Connecting a GNU/Linux Client to a Database Server	175
Setting the User File Creation Permission Mask on macOS	177
Configuring the Hosts File for a GNU/Linux Client	179
About the Location of Mount Points on GNU/Linux	180
Configuring the Shortcuts File on GNU/Linux	181
Setting a Custom Location for the Shortcuts Configuration File on GNU/Linux	182
Harmony Server Configuration	184
Using Unicode Characters in Harmony	185
Enabling Unicode Characters in Environment, Job and Scene Names	186
Enabling Unicode in Layer/Node Names	187
Configuring Scene Resolution Presets for aHarmony Database	188
Configuring Resolution Presets for Environments or Jobs	190
About Resolution Presets in XML Format	191
Chapter 3: WebCC	194
WebCC Server Installation	195
Installing WebCC on Windows	196
Disabling the Firewall for a WebCC Server on Windows	197
Installing the WebCC Service on Windows	198
Running the Database Server Service on Log-In on Windows	199
Configuring WebCC with Custom Parameters on Windows	201
About SSL, HTTPS and WebCC on Windows	206
Configuring WebCC to Use SSL on Windows	207
Setting Up Static IPs for WebCC on Windows	211
Opening the Firewall Ports for a Harmony WebCC Server on Windows	215

Checking the WebCC Server Log on Windows	217
Launching the WebCC Server Manually on Windows	218
Stopping and Restarting the WebCC Server on Windows	219
Uninstalling the WebCC Server on Windows	221
Installing WebCC on macOS	223
Installing the WebCC Server Daemon on macOS	224
Running the WebCC Server on Log-In on macOS	226
Configuring WebCC with Custom Parameters on macOS	228
About SSL, HTTPS and WebCC on macOS	233
Configuring WebCC for SSL Support on macOS	234
Configuring a macOS WebCC Server to have a Static IP Address	238
Looking Up the WebCCServer Log on macOS	241
Launching the WebCC Server Manually on macOS	242
Stopping and Restarting the WebCC Server on macOS	243
Uninstalling the WebCC Server on macOS	244
Installing WebCC on GNU/Linux	245
Disabling Security for a WebCC Server on GNU/Linux	246
Installing WebCC on GNU/Linux	248
Running the WebCC Server on Log-In on GNU/Linux	250
Configuring WebCC with Custom Parameters on GNU/Linux	252
About SSL, HTTPS and WebCC on GNU/Linux	257
Configuring WebCC for SSL Support on GNU/Linux	258
Setting Up Static IPs for WebCC on GNU/Linux	262
Opening the Firewall Ports for a WebCC Server on GNU/Linux	267
Checking the WebCC Server Log on GNU/Linux	268
Launching the WebCC Server Manually on GNU/Linux	269
Stopping and Restarting the WebCC Server on GNU/Linux	270
Uninstalling the WebCC Server on GNU/Linux	271
Configuring Render Nodes for WebCC Servers	272
Installing the Cisco OpenH264 Codec for WebCC on Windows	273

Installing the Cisco OpenH264 Codec for WebCC on macOS	274
Installing the Cisco OpenH264 Codec for WebCC on GNU/Linux	275
Configuring Local WebCC Clients	277
Configuring a GNU/Linux Workstation to Open Harmony Scenes from WebCC	278
Configuring Remote WebCC Clients	279
Disabling SSL Certificate Validation for WebCC in Harmony	280
Chapter 4: Batch Processing	282
Creating machine-list Files	284
Batch Processing Installation	285
Installing the Batch Processing Service on Windows	286
Installing the Batch Processing Service on Windows	287
Running the Batch Processing Service on Log-In on Windows	288
Looking Up the Batch Processing Log on Windows	290
Launching Batch Processing Manually on Windows	291
Stopping and Restarting the Batch Processing Service on Windows	292
Interrupting a Batch Processing Task on Windows	294
Uninstalling the Batch Processing Service on Windows	295
Installing the Batch Processing Service on macOS	297
Installing the Batch Processing Daemon on macOS	298
Running the Batch Processing Service on Log-In on macOS	300
Looking Up the Batch Processing Log on macOS	302
Launching Batch Processing Manually on macOS	303
Stopping and Restarting the Batch Processing Daemon on macOS	304
Interrupting a Batch Processing Task on macOS	305
Uninstalling the Batch Processing Daemon on macOS	307
Installing the Batch Processing Service on GNU/Linux	308
Installing the Batch Processing Daemon on GNU/Linux	309
Running the Batch Processing Service on Log-In on GNU/Linux	311
Checking the Batch Processing Log on GNU/Linux	313
Launching Batch Processing Manually on GNU/Linux	314

Stopping and Restarting the Batch Processing Daemon on GNU/Linux	315
Interrupting a Batch Processing Task on GNU/Linux	316
Uninstalling the Batch Processing Daemon on GNU/Linux	317
Setting Up Default Schedules	318
Testing Batch Processing	320
Vectorization of Scanned Drawings	321
Manually Vectorizing Scenes or Elements	322
Viewing Drawing Vectorization Lists	323
Modifying Vectorize Queue Entries	325
Rendering Scenes	326
Viewing Render Queues	327
Modifying Render Queue Entries	329
Batch Processing Schedules	330
Displaying the Schedule Status	332
Reading the Schedule Status List	334
About Default Schedules	336
Displaying Default Schedules	337
Reading Default Schedule Status Lists	339
Modifying Default Schedules	341
About Periodic Schedules	344
Displaying Periodic Schedules	345
Setting Periodic Schedules	347
Clearing Periodic Schedules	351
About Supervisory Schedules	352
Displaying Supervisory Schedules	353
Setting Supervisory Schedules	355
Clearing Supervisory Schedules	359
About Environment Shut Down	360
Shutting Down Environments	361
Starting Up Environments	363

Harmony Scheduling Utilities	365
Chapter 5: Service Launcher	366
Service Launcher Top Menu	367
Service Launcher Preferences Tab	368
Service Launcher ToonBoom Process Tab	369
Service Launcher Database Server Tab	371
Service Launcher WebCC Tab	373
Service Launcher Link Tab	375

Introduction

Harmony is a complex and full-featured animation suite that is designed for both small and large scale productions. In order to respond to the different needs of studios, Harmony is not only able to run as a stand-alone application, but also as a production server, a web server and a render farm. This guide goes through all the required steps to install and configure Harmony for all usage scenarios and on all supported platforms.

Chapter 1: Basic Installation

The following section explains how to install Toon Boom Harmony on a workstation. If you intend to use Harmony as a stand-alone application, this is the only section of the installation guide that you need. If you intend to use Harmony with a database server, you will need to refer to this section for installing the application package on your Harmony database server and its client workstations.

About Windows Installation

Installing Harmony on Windows is a straightforward process consisting mostly of running the installation wizard, then activating the product.

Installing Harmony Dependencies on Windows	14
Installing Harmony on Windows	16
Adding Harmony Executables to the %PATH% Environment Variable on Windows	18

Installing Harmony Dependencies on Windows

On Windows, some features of Harmony rely on the following software:

- **Windows Media Player** must be installed to enable Harmony's audio and video transcoding capabilities. On most consumer versions of Windows, it is installed by default and only specific editions of Windows do not have it.



NOTE

Windows Media Player is not supported on Windows 7 Starter and Windows 7 Home Basic.

- **QuickTime Player** is required to export movie files in QuickTime Movie (.mov) format as well as to import any type of movie file. Without QuickTime Player, Harmony can still export QuickTime Movie (.mov) files, but only with the H.264 (AVC) video codec and with uncompressed (PCM) audio. However, Harmony does not need QuickTime Player to launch and can be used without it.
- **Adobe Reader** is required on Windows 7 and Windows 8.1 in order to read PDF documents. It can also be installed on Windows 10, but Windows 10 is bundled with Microsoft Edge, which can be used to read PDF files.

How to install Adobe Reader

1. To install Adobe Reader, follow the instructions at the following address:

adobe.com/support/downloads/pc

How to install QuickTime Player

1. To install QuickTime Player, download and run the installer from the following address:

apple.com/quicktime/download/pc

How to install Windows Media Player

1. Windows Media Player is pre-installed on most versions of Windows, but may be disabled. Follow these steps to make sure it is enabled.

- a. Open the Windows Control Panel:
 - On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
 - On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.
 - b. From the Control Panel, go to **Programs > Turn Windows features on or off**.
 - c. In the list of Windows features, find and expand the **Media Features** option. **Windows Media Player** should be listed under Media Features. If either of these options are not listed, see below.
 - d. Ensure both the Media Features and Windows Media Player check boxes are checked.
 - e. Click **OK**.
 - f. Wait for the installation to finish.
 - g. Restart Windows.
2. If Media Features or Windows Media Player is not listed in the available features, make sure your edition of Windows supports it. If it does, it may be added by installing the Media Feature Pack for Windows. To do this, click on the link corresponding to your version of Windows and follow the instructions:
- **Media Feature Pack for Windows 7:**
<https://www.microsoft.com/download/details.aspx?id=16546>
 - **Media Feature Pack for Windows 8:**
<https://www.microsoft.com/download/details.aspx?id=30685>
 - **Media Feature Pack for Windows 10:**
<https://www.microsoft.com/download/details.aspx?id=48231>

Installing Harmony on Windows

You can install Harmony on Windows by running the installation wizard. Once the installation process is finished, you will need to activate Harmony before you can use it.



NOTE

You need an account with administrator privileges to launch the installer.

How to install Harmony on Windows

1. In a file browser, double-click on the installer package you have downloaded.
2. When prompted by Windows, give the permission to the installer to make changes to your device.



NOTE

If you are not currently logged in with an administrator account, you will need to provide the username and password of an account with administrator privileges.

3. In the **Choose Setup Language** prompt, select the preferred language for the installation process and for reading the end-user license agreement.
4. Wait for the installer to prepare the installation.
5. In the welcome screen, click on **Next**.
6. If you accept the license agreement presented in the installation wizard, select **I accept the terms in the license agreement**, then click on **Next**. Otherwise, click on **Cancel**.
7. If you want to install Harmony in the default installation directory, select **Next**. Otherwise, click on **Change** and select a preferred installation directory.
8. In the Installation Type dialog, select the appropriate package type:
 - If you intend to work in a Harmony Server environment, whether as server or as a client workstation, select **Complete**.
 - If you only intend to work in a fully standalone environment or if you only intend to connect to a Harmony database using WebCC only, select **Standalone**.
9. Click on **Install**.
10. Wait for the installation process to finish. Once the process is finished, you are ready to launch the application.
11. Open the Windows Start menu.
12. In the Programs list, select **Harmony 17 [Edition] > Harmony [Edition]**.

13. If your product is not activated, the Toon Boom Activation Wizard will launch. For information on how to activate Harmony, see the Toon Boom License Activation Documentation at the following address:

<https://docs.toonboom.com/help/activation>

Adding Harmony Executables to the %PATH% Environment Variable on Windows

You can add the path to the Harmony executable files to the **%PATH%** environment variable. This will allow you to run Harmony and its applications and utilities from the command line by typing the name of the executable files, without having to type their full path.



NOTE

You need an account with administrator privileges to do this.

How to add Harmony executables to the %PATH% environment variable on Windows

1. In the Start menu, open the Programs menu and select **Harmony 17 [Edition] > Tools > Configuration Wizard**.
2. When prompted, enable Configuration Wizard to make changes to your device.
3. In the Toon Boom Harmony Configuration dialog, select the ... button right of **Add Harmony to Path**.
4. If you only want to add the Harmony executables to the **%PATH%** variable for your own user, select **Single User**. If you want the Harmony executables to be in the **%PATH%** variable regardless of who is logged in, select **All Users**.
5. Select **Next**.
6. Click on **Finish**.
7. To verify that the change has been applied, open a new Command Prompt window and type the following command:

```
> echo %PATH%
```

The path to the Harmony **bin** folder should be included in the output, separated by other paths with a semi-colon.

About macOS Installation

On macOS, Harmony is provided in an Apple Disk Image (**.dmg**) file, which contains the software package. The installation procedure mostly consists of copying its content to the Applications directory of your macOS computer.

Installing Harmony on macOS	20
Launching Harmony and Side Applications on macOS	21
Adding Harmony Binaries to the \$PATH Environment Variable on macOS	22

Installing Harmony on macOS

To install Harmony on macOS, you must open the Apple Disk Image (.dmg) file, then copy its content to the Applications folder on your machine. You will need administrator privileges to be able to add applications to the Applications folder.

How to install Harmony on macOS

1. In a file browser, double-click on the application package you have downloaded.

Then the License Agreement prompt appears.

2. If you accept the license agreement presented by the application package, click on the **Agree** button. Otherwise, click on **Disagree**.



TIP

If you want to see the license agreement in another language, select a language in the language drop-down above the license agreement.

3. Wait for macOS to verify the content of the package.
4. In the Finder window that opens, select **H Toon Boom Harmony 17 [Edition]**.
5. Drag and drop the selected folder over **Applications**.



NOTE

If you are not logged-in as an administrator user, you will be prompted by macOS to enter the log-in and password of a user with administrator privileges to authorize making changes to the Applications folder.



6. Wait for the package to finish copying.

Once the package is done copying, Harmony is installed.

Launching Harmony and Side Applications on macOS

Harmony is packaged with several side applications. However, in order to comply with the security requirements of macOS, Harmony is packaged like a single application since Harmony 17. In order to be able to launch its side applications, you must first launch Harmony, then select a side application from the top menu.



How to launch a Harmony side application when using Harmony Stand Alone

1. In Finder, proceed to **Applications >  Toon Boom Harmony 17 [Edition]**.
2. Double-click on  **Harmony [Edition]**.


The main Harmony application launches and the Welcome dialog appears.

3. To launch **License Wizard, Play, Configuration Assistant** or **Configuration Editor**, open the **Harmony [Edition]** top menu, select **Tools** and, in the sub-menu, select the application you want to launch.

How to launch a Harmony side application when using Harmony Server

1. In Finder, proceed to **Applications >  Toon Boom Harmony 17 [Edition]**.
2. Double-click on  **Harmony [Edition]**.


The main Harmony application launches and the log-in dialog appears.

3. Do one of the following:
 - If you want to launch Control Center, simply click on the  Control Center button in the bottom-left corner of the log-in dialog.
 - If you want to launch any other side application, such as **Paint, Play, Scan, Configuration Assistant, Configuration Editor, License Wizard** or **Service Launcher**, open the **Harmony [Edition]** top menu, select **Tools** and, in the sub-menu, select the application you want to launch.

Adding Harmony Binaries to the \$PATH Environment Variable on macOS

You can add the path to the Harmony binary files to the `$PATH` environment variable. This will allow you to run Harmony and its applications and utilities from a terminal by typing the name of the executable files, without having to type their full path.

How to add Harmony binaries to the \$PATH environment variable on macOS

1. In Finder, proceed to **Applications >  Toon Boom Harmony 17 [Edition]** and double-click **Harmony [Edition]**.
The main Harmony application launches.
2. In the top menu, select **Harmony [Edition] > Tools > Configuration Assistant**.
3. In the Welcome screen, check the **Register console applications in the path** option and uncheck all other options.
4. Click on **Continue**.
5. If you only want to add the Harmony binaries to the `$PATH` variable for your own user, select **Register Path for my user only**. If you want the Harmony binaries to be in the `$PATH` variable regardless of who is logged in, select **Register Path for all users**.



NOTE

You need an account with administrator privileges to register the path for all users.

6. Click on **Create**.

If you selected Register Path for all users, you will be prompted to enter an administrator's username and password to authorize the change.

7. To verify that the change has been applied, open a new Terminal window and type the following command:

```
$ echo $PATH
```

The path to the Harmony `bin` folder should be included in the output, separated by other paths with a colon.

About GNU/Linux Installation

On GNU/Linux, Harmony is distributed as an application package in a compressed archive and is bundled with an installation script. Just like a regular installer, the installation script can install the package and add shortcuts to the applications to your desktop environment, but it is also able to perform the basic configuration routines if you want to install a database or WebCC server or to set-up a batch processing node.

Harmony is built to support the following operating systems:

- CentOS 6 / Red Hat Enterprise Linux 6
- CentOS 7 / Red Hat Enterprise Linux 7 (up to 7.3).

The installation instructions will presume you are working on a fresh installation of one of these operating systems.

If you attempt to install Harmony on another GNU/Linux-based operating system, keep in mind that:

- This is not a supported scenario.
- Harmony may crash when proceeding to the Welcome screen or main application window. This issue is known to occur on later versions of CentOS 7 (starting with 7.4) as well as of Fedora 23 and later. It may be possible to work around this issue—see [Troubleshooting a Crash When Loading Harmony on GNU/Linux on page 42](#).
- Harmony may fail to launch and it may be impossible to work around the issue.
- Harmony's daemons may not be compatible with the daemon system on your operating system, which means the use of Service Launcher or the creation of custom-made launch scripts may be required to set up a database server, a WebCC server or a batch processing node.

About GNU/Linux Installation

When you install GNU/Linux, you have several options as to the software packages that come pre-installed with it. To avoid dependency issues, you should install GNU/Linux with a desktop environment and a full suite of desktop software. More specifically:

- When installing **CentOS 6** or **Red Hat Enterprise Linux 6**, you should install the **Desktop** or **Software Development Workstation** software set.
- When installing **CentOS 7** or **Red Hat Enterprise Linux 7**, you should install the **GNOME Desktop** or **Development and Creative Workstation** base environment.



IMPORTANT

It is highly recommended to install the desktop software suite for your GNU/Linux distribution even if you are installing GNU/Linux for a Harmony database server or a license server, as none of the applications bundled with Harmony have been tested to work in a minimal environment on GNU/Linux.

Installing Harmony Dependencies on GNU/Linux

On GNU/Linux, Harmony requires the following packages to be installed:

- `SDL`
- `libX11`
- `libXext`
- `libXrandr`
- `alsa-lib`
- `libvorbis`
- `libSM`
- `fontconfig`
- `libGLU`
- `libjpeg`
- `libpng`
- `libXi`
- `tcsh`
- `redhat-lsb`

If you have a GNOME Desktop-type installation, most of these will be already installed, but at least some of these will usually need to be installed manually. If you have a minimal installation, all of these packages will need to be installed manually.

How to install the Harmony dependencies on GNU/Linux

1. Open a terminal and type in the following command.

```
$ sudo yum install SDL libX11 libXext libXrandr alsa-lib libvorbis  
libSM fontconfig libGLU libjpeg libpng libXi tcsh redhat-lsb
```

2. When prompted, confirm that you want to proceed with the installation.

Installing Proprietary Video Drivers on GNU/Linux

Harmony requires that the official (*proprietary*) drivers for your video card are installed in order to properly use OpenGL to preview your artwork and animation. However, CentOS and Red Hat Enterprise Linux come with a free and open source adaptation of the driver for your graphic card, with which Harmony is likely to encounter issues.

Some video drivers come as application packages that are already made to install on your system without a hassle. In this case, all you need to do is obtain the driver from your video card chipset manufacturer and follow the installation instructions. Some other video drivers are distributed as scripts that build the drivers into kernel modules. These are more complicated to install and this topic will break down the process, which consists of the five following steps:

1. Obtain the official video driver from your video card chipset manufacturer's website.
2. Install the utilities required to build the video driver as a kernel module.
3. Disable the default video driver bundled with your operating system.
4. Launch GNU/Linux in terminal mode.
5. Run the official video driver's installation script.

How to know the model of your video card

1. Open a terminal and type in the following command:

```
$ lspci | grep 'VGA compatible controller'
```

The utility will output a line in this format:

```
01:00.0 VGA compatible controller: <video card model>
```

With this information, you can search for the website of the chipset manufacturer on the Internet, and obtain the proper drivers from it.

How to install the utilities required to build the video driver as a kernel module

1. Open a terminal and type in the following command:

```
$ sudo yum install "kernel-devel-uname-r == $(uname -r)" || sudo yum  
install kernel-devel
```

This command will install the headers for your Linux kernel, which are required to build a kernel module such as a video card driver. It tries the command for installing kernel headers for an old kernel and, if that fails, the command for installing kernel headers for the latest version of the kernel, to cover both cases.

2. Enter the following command to install the remaining dependencies:

```
$ sudo yum install kernel-headers gcc make dkms acpid
```

How to disable the default drivers on GNU/Linux

1. First, we must obtain the name of the kernel module used by your video card. To do that, start by getting the address used by your video card with this command:

```
$ lspci | grep 'VGA compatible controller'
```

The output will be in this format:

```
01:00.0 VGA compatible controller: <video card model>
```

The numbers before **VGA compatible controller** are the address of the video card. They should be in the **##:##.#** format. Make note of them.

2. Search for the module associated with this address by entering the following command. Replace **##:##.#** with the address used by your video card.

```
$ find /sys/bus/pci/drivers -name "????:##:##.#"
```

The output will look like this:

```
/sys/bus/pci/drivers/nouveau/0000:01:00.0
```

The name of the folder just under **drivers** is the name of the kernel module. In this example, it is **nouveau**, which is typically used for NVIDIA graphic cards.

3. Open the list of blacklisted kernel modules in a text editor:

```
$ sudo vi /etc/modprobe.d/blacklist.conf
```

4. Add the name of the kernel module for your video card to the list of blacklisted modules:

```
blacklist <name of kernel module>
```

For example, if the kernel module for your video card is **nouveau**, you would add the following line:

```
blacklist nouveau
```

5. Rebuild **initramfs**. With the default video driver added to the blacklist, **initramfs** will be configured to exclude these drivers from the list of modules it loads on boot, and load the basic video driver instead:

```
$ sudo mv /boot/initramfs-$(uname -r).img{,~}
$ sudo dracut -v /boot/initramfs-$(uname -r).img $(uname -r)
```

6. Reboot the machine.

**NOTE**

After rebooting, it is possible that your display resolution will no longer be optimal. This is a sign that you successfully disabled the default video driver. Your display resolution will be optimal again after you install the proprietary video driver.

7. After logging in, you can verify that the default video drivers are no longer loaded by running the following command.

```
$ lsmod | grep "<name of kernel module>"
```

For example, if the kernel module for your video card was nouveau, you could use the following command:

```
$ lsmod | grep "nouveau"
```

If it returns nothing, GNU/Linux is now running on basic video drivers that will not interfere with the installation of proprietary drivers.

How to launch GNU/Linux in terminal mode and run the driver installation utility

1. Reboot your machine.
2. At the GNU GRUB boot menu, use the Up and Down arrow to move the cursor to the boot menu entry that you usually use to boot GNU/Linux.
3. Press the E key.

GRUB opens the selected entry in a text editor, allowing you to temporarily modify its parameters for this boot cycle only.

**NOTE**

The changes you make to the boot menu entry will only affect this launch and will not be preserved next time you boot the machine.

4. Using the Up and Down key, move the cursor to the line that starts with **kernel** or **linux16**.
5. Use the End key to move the cursor to the end of that line.
6. Add a space at the end of the line, followed by **3**. This is a boot parameter that will instruct GNU/Linux to boot at run level 3, which means it will load a simple terminal with networking capabilities.
7. Press Ctrl + X.

GNU/Linux will begin booting. At the end of the process, you will be presented with a simple terminal and prompted to log in by typing your username and password.

8. Log in with the username and password of a user with administrative privileges.

9. Navigate to the directory in which you downloaded the driver installation utility.

```
$ cd ~/Downloads
```

10. Add execute permissions to the utility.

```
$ chmod +x ./path_to_driver_installer.run
```

11. Run the utility.

```
$ sudo ./path_to_driver_installer.run
```

12. Follow the instruction to install the driver. If the installation is successful, the driver will be loaded next time you boot GNU/Linux.

Installing Harmony on GNU/Linux

To install Harmony on GNU/Linux, you must extract the package from the .tar.gz archive you downloaded, then run the installation script it contains.

You can choose to run the installation script interactively or with command line parameters to automate most of the process.



NOTES

- If you want to run the installation script interactively, your operating system must have the **whiptail** command line tool, which is provided with the **newt** application package.
- Even if you install Harmony non-interactively, you must still interactively review and agree to the license agreement on each installation.

How to install Harmony on GNU/Linux interactively

1. Verify that **whiptail** is present on your system, to allow the installer to use interactive mode:

```
$ sudo yum install newt
```

2. In a terminal, navigate to the directory where you downloaded the installation package.

```
$ cd ~/Downloads
```

3. Extract the package's contents in the directory.

```
$ tar -xvzf ./name-of-package.tar.gz
```

4. Switch to the directory in which the package's contents were extracted.

```
$ cd name-of-package
```

5. Run the install script.

```
$ sudo ./install
```

The interactive setup launches in the terminal.

6. Do one of the following:
 - If you are working in a Harmony Server environment, whether your machine is going to be a client workstation or a server, select **SERVER** and press Enter.
 - If you are working in a fully standalone environment or if you only intend to connect to a Harmony database using WebCC, select **STANDALONE** and press Enter.
7. In the components dialog, for each component you want to install, use the Up and Down keys to highlight the component, and the Spacebar key to check or uncheck the component:

- **APPLICATION:** Installs the application files: binaries, libraries, resources, configuration files, etc. This is required to be able to launch Harmony. You should only omit this option if you are using the installer to add components to an existing installation. .
- **DESKTOP:** Creates a directory in the Applications menu of your desktop environment and adds a shortcut for each Harmony application inside it. This option also allows you to open a scene in Harmony by double-clicking on an **.xstage** file in a file browser.
- **PATH:** Adds the path to the Harmony binaries to your \$PATH environment variable, allowing you to launch Harmony applications from a terminal by typing their name instead of their full path.
- **URL:** Registers the harmony:// URL scheme so that such URLs launch Harmony. This allows WebCC clients to launch Harmony directly from the web browser.

If you selected to install Harmony Server, these options will also be available:

- **PROCESS:** Enables the batch processing daemon and adds the current machine to the database's list of machines available for batch processing.

**NOTE**

Further configuration is required to make this machine a working batch processing node.

- **DATABASE:** Enables the database server daemon and, if no database is present on the machine, creates an empty database file system on the local machine.
 - **LINK:** Enables and configures the link server daemon. This is required if the current machine is meant to be a database server for Windows clients.
 - **WEBCC:** Enables and configures the WebCC daemon. This will make this machine a WebCC server.
 - **OPENH264:** Downloads and installs the Cisco OpenH264 library. This allows a WebCC server to create preview videos for the database scenes which can be played in a web browser. An Internet connection is required.
 - **UPDATEDB:** Replaces the dictionary (.dict) files in this machine's database with this package's dictionary files. This is useful if your machine is a database server, and you are currently upgrading its version of Harmony.
8. Press the **Tab** key to switch to the Ok button at the bottom, then press **Enter**.
 9. In the Installation Directory dialog, you can choose to type a custom installation path, or to use the default one. Then, press **Enter**.

10. In the License Agreement dialog, take the time to carefully review the license agreement. Use the **Up** and **Down** keys to scroll through the text in the agreement and read it until the end.
11. Press **Tab** to switch to the AGREE and DISAGREE buttons. You can use the **Left** and **Right** keys to switch between selecting the AGREE or the DISAGREE button:
 - If you agree with the license agreement, select the **AGREE** button and press **Enter**.
 - If you disagree with the license agreement, select the **DISAGREE** button and press **Enter**.
12. A dialog will prompt you to confirm each component that you elected to install. If a lot of components are listed, you can use the **Up** and **Down** keys to scroll through the list. You can use the **Left** and **Right** keys to select Yes or No. If you want to proceed, select **Yes** and press **Enter**.
13. The installation begins. Each step of the installation is explained in detail until the installation is over.
14. If your product is not activated, do one of the following to launch the License Wizard:
 - In the terminal, type the following command:


```
$ sudo /usr/local/ToonBoomAnimation/harmony[Edition]_17/lnx86_64/bin/LicenseWizard
```
 - In the GNOME top menu, select **Applications > Harmony 17 [Edition] > License Wizard**.
A terminal will open, prompting you to enter your user password. This terminal is attempting to launch License Wizard with sudo, as it requires elevated privileges. Type in your password and press Enter.

For information on how to use the License Wizard to activate Harmony, see the Toon Boom License Activation Documentation at the following address:

<https://docs.toonboom.com/help/activation>

15. Once Harmony is installed and activated, you can start using it. However, if you elected to add the path to Harmony's binaries to your **\$PATH** environment variable, you will need to reboot your machine before you can benefit from this feature.

How to install Harmony on GNU/Linux non-interactively

1. In a terminal, navigate to the directory where you downloaded the installation package.

```
$ cd ~/Downloads
```

2. Extract the package's contents in the directory.

```
$ tar -xvzf ./name-of-package.tar.gz
```

- Switch to the directory in which the package's contents were extracted.


```
$ cd name-of-package
```


- Start typing the command to run the installer with all the parameters you want for your installation:

```
$ sudo ./install <parameters>
```



**NOTE**

Each parameter must be specified separately. For example: **-a -p** will work, but not **-ap**.

Parameter	Description
Installation modes	
-server	Installs the application with shortcuts to the server applications and with daemon files for database servers and clients. <div data-bbox="630 940 1403 1077" style="border: 1px solid green; padding: 5px; margin-top: 10px;">  <p>NOTE If no installation mode is specified, this one is implied by default.</p> </div>
-standalone	Installs the application without the shortcuts to the server applications and without the daemon files for database servers and clients. Only use this option if you intend to use Harmony on this machine without relying on a Harmony Server database.
-uninstall	Uninstalls the specified components instead of installing them. If specified alone, this launches the uninstaller in interactive mode.
Optional components	
-a -application	Installs the application files: binaries, libraries, resources, configuration files, etc. This is required to be able to launch Harmony. You should only omit this option if you are using the installer to add components to an existing installation. .
-b	For backward compatibility with older versions of the installer, this installs the application files and adds the path to the Harmony binaries to

Parameter	Description
-binaries	the \$PATH environment variable. It is equivalent to -a -e .
-de -desktop -kde	Creates a directory in the Applications menu of your desktop environment and adds a shortcut for each Harmony application inside it. This option also allows you to open a scene in Harmony by double-clicking on an .xstage file in a file browser.
-d -dbserver	Enables the database server daemon and, if no database is present on the machine, creates an empty database file system on the local machine.
-l -linkserver	Enables and configures the link server daemon. This is required if the current machine is meant to be a database server for Windows clients.
-p -process	Enables the batch processing daemon and adds the current machine to the database's list of machines available for batch processing. <div data-bbox="630 982 1403 1155" style="border: 1px solid green; padding: 10px; margin-top: 10px;">  NOTE Further configuration is required to make this machine a working batch processing node. </div>
-w -webccserver	Enables and configures the WebCC daemon. This will make this machine a WebCC server.
-oh -openh264	Downloads and installs the Cisco OpenH264 library. This allows a WebCC server to create preview videos for the database scenes which can be played in a web browser. An Internet connection is required.
Configuration switches	
-e -env	Adds the path to the Harmony binaries to your \$PATH environment variable, allowing you to launch Harmony applications from a terminal by typing their name instead of their full path.

Parameter	Description
-s -schemeregister	Registers the harmony:// URL scheme so that such URLs launch Harmony. This allows WebCC clients to launch Harmony directly from the web browser.
-u -updatedict	Replaces the dictionary (.dict) files in this machine's database with this package's dictionary files. This is useful if your machine is a database server, and you are currently upgrading its version of Harmony.
Other options	
-h -help	Displays information about how to use the install script.
-target <PATH>	Specifies the directory in which to install the application. If not specified, Harmony will be installed in /usr/local/ToonBoomAnimation/harmony [Edition]_17
-dbpath <PATH>	Specifies the location of the database on this machine's file system. This option is only relevant if installing a database server with the -d option, installing a WebCC server with the -w option or updating dictionary (.dict) files with the -u option. If not specified, the installer will presume that the database is located in /USA_DB .
-own -own <USER> -own <USER:GROUP>	Makes the application files owned by the specified user or by the specified user:group. If no user or user:group is specified, this will make the application files owned by the user running the script. If this option is not specified, application files will be owned by the root user.
-q -quiet	Do not output information during the installation process.
-v -verbose	Output detailed information on each step of the installation process, including every file copied.
-output <PATH>	Writes the installation log to the specified file instead of outputting it to

Parameter	Description
	the terminal.
-nodialog	<p>Disables the use of the whiptail utility to display dialogs and interactive prompts. This mode is compatible with <i>dumb</i>-type terminals, which do not support scrolling or erasing the screen.</p> <div style="border: 1px solid green; padding: 5px;">  <p>NOTE This option is automatically enabled if the whiptail utility is not found on the machine or if the installer detects that the terminal is unable to scroll or erase the screen.</p> </div>
-force	<p>By default, installing over an existing installation will simply overwrites existing files, except for configuration files in the etc directory of the installed application, which the installer preserves. This option will cause the existing installation to be completely deleted before installing.</p> <div style="border: 1px solid red; padding: 5px;">  <p>IMPORTANT Enabling this option will cause your Harmony configuration files to be erased permanently.</p> </div>
-y -assumeeyes	Assumes yes on all prompts, except for the license agreement.

5. Press Enter.

The install script launches and, if the parameters are valid, the License Agreement prompt appears.

6. In the License Agreement dialog, take the time to carefully review the license agreement. Use the **Up** and **Down** keys to scroll through the text in the agreement and read it until the end.
7. Press **Tab** to switch to the AGREE and DISAGREE buttons. You can use the **Left** and **Right** keys to switch between selecting the AGREE or the DISAGREE button:
 - If you agree with the license agreement, select the **AGREE** button and press **Enter**.
 - If you disagree with the license agreement, select the **DISAGREE** button and press **Enter**.
8. A dialog will prompt you to confirm each component that you elected to install. If a lot of components are listed, you can use the **Up** and **Down** keys to scroll through the list. You can use the **Left** and

Right keys to select Yes or No. If you want to proceed, select **Yes** and press **Enter**.

9. The installation begins. Each step of the installation is explained in detail until the installation is over.
10. If your product is not activated, do one of the following to launch the License Wizard:
 - In the terminal, type the following command:

```
$ sudo /usr/local/ToonBoomAnimation/harmony[Edition]_17/lnx86_64/bin/LicenseWizard
```

- In the GNOME top menu, select **Applications > Harmony 17 [Edition] > License Wizard**.

A terminal will open, prompting you to enter your user password. This terminal is attempting to launch License Wizard with sudo, as it requires elevated privileges. Type in your password and press Enter.

For information on how to use the License Wizard to activate Harmony, see the Toon Boom License Activation Documentation at the following address:

<https://docs.toonboom.com/help/activation>

11. Once Harmony is installed and activated, you can start using it. However, if you elected to add the path to Harmony's binaries to your **\$PATH** environment variable, you will need to reboot your machine before you can benefit from this feature.

Adding Harmony Binaries to the \$PATH Environment Variable on GNU/Linux

You can add the path to the Harmony binary files to the **\$PATH** environment variable. This will allow you to run Harmony and its applications and utilities from a terminal by typing the name of the executable files, without having to type their full path.

How to add Harmony binaries to the \$PATH environment variable on GNU/Linux

1. In a Terminal, navigate to the location where you previously extracted the Harmony package for installation—see [Installing Harmony on GNU/Linux](#).

```
$ cd ~/Downloads/name-of-package
```

2. Run the installation script with the **-e** option.

```
$ sudo ./install -e
```

The Toon Boom Harmony license agreement prompt appears.

3. In the License Agreement dialog, take the time to carefully review the license agreement. Use the **Up** and **Down** keys to scroll through the text in the agreement and read it until the end.
4. Press **Tab** to switch to the AGREE and DISAGREE buttons. You can use the **Left** and **Right** keys to switch between selecting the AGREE or the DISAGREE button:
 - If you agree with the license agreement, select the **AGREE** button and press **Enter**.
 - If you disagree with the license agreement, select the **DISAGREE** button and press **Enter**.

5. A dialog will prompt you to confirm that you want to add the path to the Harmony binaries to the **\$PATH** environment variable. You can use the Left and Right arrow keys to select Yes or No. Select **Yes** and press Enter.

The install script will add the path to the Harmony binaries to the **\$PATH** environment variable.

6. To verify that the change has been applied, open a new Terminal window and type the following command:

```
$ echo $PATH
```

The path to the Harmony **bin** folder should be included in the output, separated by other paths with a colon.

About Audio Transcoding on GNU/Linux

On GNU/Linux, by default, Harmony can only play audio files with these specifications:

- **Format:** Waveform Audio File (.wav)
- **Sample rate:** 22 050 Hz
- **Sample size:** 16 bits
- **Channel mode:** Stereo

If FFmpeg is installed on your system, Harmony will use it to transcode other audio formats into this format on the fly. However, FFmpeg comes bundled with patented algorithms such as encoders and decoders, the unlicensed use of which may contravene intellectual property laws in your jurisdiction. Hence, if you use Harmony on GNU/Linux, it is recommended to transcode your audio in the format specified above using third-party software before importing it to your Harmony scene.

Preventing the Alt Key Keyboard Shortcut Conflict Between Harmony and the GNU/Linux Desktop

By default, in GNOME 2 and KDE, clicking anywhere on a window while holding the Alt key down will move the window. Because the Alt key is used as a modifier for some drawing and animations tools in Harmony, this may come in conflict with Harmony functionalities. Hence, it is highly recommended to configure your desktop environment to use the *Super* (Windows) key for moving windows instead.

How to change the window moving keyboard shortcut in GNOME 2

1. In the GNOME top menu, select **System > Preferences > Windows**.
The Window Preferences dialog appears.
2. In the **Movement Key** section, select **Super (or "Windows Logo")**
3. Click on **Close**.

How to change the window moving keyboard shortcut in KDE

1. In the KDE menu, select **Computer > System Settings**.
2. In the System Settings window, select the **Windows Behavior** icon.
3. Select the **Window Actions** tab.
4. In the **Modifier Key** drop-down, select **Meta**.
5. Click on **Apply**.

Troubleshooting Problems with Harmony on GNU/Linux

The following section explains how to fix common issues with Harmony on GNU/Linux.

Troubleshooting a Crash When Loading Harmony on GNU/Linux

If Harmony crashes when opening a scene or proceeding to the offline welcome screen, returning the message "**segmentation fault**" to the terminal, this may be caused because Harmony is not compatible with later versions of OpenSSL. You can fix this by setting the `OPENSSL_ia32cap` environment variable to the value "`~0x2000002000000000`". This can be done in three different ways:

- Setting the environment variable in a terminal and launching Harmony from this terminal.



TIP

You should attempt this first to make sure that setting this environment variable actually solves the problem.

- Setting the environment variable for one user, and launching Harmony when logged in as this user.
- Setting the environment variable for the entire machine, allowing you to launch Harmony as any user. This requires an account with administrator privileges.

How to set the `OPENSSL_ia32cap` environment variable in a terminal and launch Harmony

1. Open a terminal and enter the following command:

```
$ export OPENSSL_ia32cap=~0x2000002000000000"
```

2. Launch Harmony from the same terminal:

```
$ /usr/local/ToonBoomAnimation/harmony[Edition]_17/lnx86_64/bin/Harmony[Edition]
```

How to set the `OPENSSL_ia32cap` environment variable for the current user

1. Open a terminal and open your Bash shell login file:

```
$ vi ~/.profile
```

2. Add the following line at the end:

```
export OPENSSL_ia32cap=~0x2000002000000000"
```

3. Save and quit.

4. Open your C-shell login file:

```
$ vi ~/.login
```

5. Add the following line at the end:

```
setenv OPENSLL_ia32cap "~0x200000200000000"
```

6. Save and quit.
7. Log out, then log back in.
8. Attempt to launch Harmony.

How to set the OPENSLL_ia32cap environment variable for all users

1. Create a new Bash boot script using the following command:

```
$ sudo vi /etc/profile.d/openssl.sh
```

2. In the script, type the following command:

```
export OPENSLL_ia32cap=~0x200000200000000"
```

3. Save and quit.
4. Create a new C-shell boot script using the following command:

```
$ sudo vi /etc/profile.d/openssl.csh
```

5. In the script, type in the following command:

```
setenv OPENSLL_ia32cap "~0x200000200000000"
```

6. Save and quit.
7. Reboot the machine.
8. Attempt to launch Harmony.

Troubleshooting Desktop Shortcuts Launching the Wrong Version of Harmony

If you have several older versions of Harmony installed, it is possible that some of their shortcuts launch the wrong of Harmony. This is because, prior to Harmony 15.0, desktop shortcuts created for Harmony would not point to the full path of applications, so they launched the version of the application in the `$PATH` environment variable.

A script has been made to fix this issue. It will go through every installation of Harmony in the default installation location and create shortcuts for them with the proper configuration.

You can download the script from the following location:

https://docs.toonboom.com/download/tools/fix_harmony_shortcuts.sh



NOTE

This script will only work for installations of Harmony that are located in the default directory:
`/usr/local/ToonBoomAnimation`.

How to launch the desktop shortcut fixing script

1. In a terminal, go to the directory where you downloaded the script.

```
$ cd <download directory>
```

2. Add execution privileges to the script.

```
$ chmod +x fix_harmony_shortcuts
```

3. Run the script with elevated privileges:

```
$ sudo ./fix_harmony_shortcuts
```

4. Read the confirmation prompt, then type **yes** and press enter.

Chapter 2: Harmony Server

Harmony Server is a team-based infrastructure for animation production. Its array of cutting-edge features increases productivity and encourages creativity. Developed in conjunction with leading studios worldwide, Harmony is specifically designed for long-term episodic and feature length projects, providing a true animation pipeline. Several animators can work simultaneously on the same scene while the asset library provides users with easy access to up-to-date media assets.



Harmony Server is truly scalable; more than 100 staging clients can share animation just as easily as a few can. Whether you are a start-up studio or a large established animation facility, Harmony serves as the animation backbone ensuring sustainability and growth.

Harmony Server is a powerful solution that brings together multiple teams working on the same project, whether in-house or remotely. The Harmony Serversolution offers a robust asset management system that enables users to quickly locate assets, share tasks on complex scenes and centralize all assets in a common repository. Tremendous gains in efficiency and quality are made by the teams, who enjoy a smooth flow between each task, and more time dedicated to their creative assignments.

In addition, the server is critical at the end of a production for rendering projects efficiently. The rendering process, controlled directly by the server, not only renders production scenes but also executes the batch vectorization of the scanned drawings. You can send an unlimited number of scenes to render and change their priorities depending on your deadline. You can also render final frames locally or batch process over a network in the background or at scheduled times. For more information, see [Batch Processing on page 282](#).

About the Harmony Server Structure

The primary purpose of the database server is to host and manage the information on its Harmony scenes. Optionally, it can also host the actual scene data, but other machines on your network can be used for scene data storage as well, as those are two different components of a Harmony database.

A Harmony database is structured using different network shares:

- The **/USA_DB** share, which contains the information on all the scenes, environments, jobs as well as shared assets such as palettes and templates stored in environments or jobs. This part of the database must be hosted on the machine that runs the database server service/daemon, to which client workstations will connect directly.
- **/usadata** shares, which contain the actual scene data: the drawings, animation, 3D models, scene palettes, etc. Those can be hosted on the database server or on any machine on the local network that the client workstations can access. There can be as many as the studio needs, which allows you to expand the storage capacity of your database server by creating a new **/usadata** share on a new file server.

On client machines, every network share is mounted on the same location as the location of the actual share on their respective server. Likewise, if other servers are used to store scene data, the shares of these servers are mounted on the database server. The result is that the database file structure is entirely accessible on all machines in the network.

If you intend to use several **/usadata** shares, they must be numbered them. The best practice is to name the first **/usadata** share **/usadata000**, and it is typical, although not mandatory, to store that share on the same machine as the database server. Then, when you need to expand the storage space for your Harmony database, you can create a share named **/usadata001** on another machine, and another one named **/usadata002** on another machine, etc.

In the end, client machines will locate the different shares that compose your database by mounting those share points on the client machine's filesystem root.

On Windows machines, this is done by adding shortcut (.lnk) files in the root of a drive (typically C:) which point to the network path of the share, for example:

- A **USA_DB.lnk** shortcut file pointing to **\\dbserver\USA_DB**.
- A **usadata000.lnk** shortcut file pointing to **\\dbserver\usadata000**.
- A **usadata001.lnk** shortcut file pointing to **\\newfileserver\usadata001**.

This process is automatically done by Configuration Wizard, a Windows-exclusive configuration tool for Harmony.

On macOS and GNU/Linux machines, this is done by mounting the shares on folders located at the root of the filesystem. By giving those folders the name of their respective shares, it simulates the presence of the server's files on the machine's filesystem. For example:

- A directory named `/USA_DB` is mounted with the NFS share at address `dbserver:/USA_DB`.
- A directory named `/usadata000` is mounted with the NFS share at address `dbserver:/usadata000`.
- A directory named `/usadata001` is mounted with the NFS share at address `newfileserver:/usadata001`.

Typically, clients and servers can communicate without any issue if they use the same kind of operating system. It is possible to set-up a Harmony database that mixes different platforms across the database server, the file servers and the client workstations, but with some important limitations that must be taken in account:

- By default, the use of non-alphanumerical Unicode characters, with the exception of the dash and the underscore, in Harmony for naming scenes, layers and drawings is not enabled, but can optionally be enabled. However, you must absolutely keep in mind that macOS and GNU/Linux typically fail to communicate filenames containing Unicode characters over the NFS protocol. **This means that if Unicode is enabled on your database, and that a macOS based client tries to open or save a scene containing Unicode characters which is hosted on a GNU/Linux-based file server, Harmony will fail to open and save files with Unicode characters in their name, and there might be data loss!** Hence, it is important to never enable Unicode if your network includes macOS-based clients and GNU/Linux-based database servers or file servers, including GNU/Linux-based NAS servers.
- If you set up your database server on a Windows machine, only Windows clients are supported. It may be possible to connect a macOS-based or GNU/Linux-based client workstation to a Windows-based database server, but this may cause unexpected issues and it is not a supported usage scenario.
- If you intend to set up your database server on a macOS machine and you want Windows workstations to be able to connect to it, you will need to install a third party software, SMBup, and configure it to share your database files through the Samba protocol. This is because macOS natively supports the NFS sharing protocol, but not SMB, and Windows natively supports the SMB file sharing protocol, but not NFS.

To summarize, here is a table outlining the implications of mixing platforms between servers and clients:

↓Client Server →	Windows	macOS	GNU/Linux
Windows	No problem.	Requires SMBup to be installed and configured on the server.	No problem.
macOS	Not supported.	No problem.	Unicode must be disabled.
GNU/Linux	Not supported.	Unicode must be disabled.	No problem.

About Database Server Installation

At the heart of the Harmony Server solution is the server itself, which centralizes all the production assets in a repository. Loaded with production proven tools to manage administration tasks, the server is completely flexible and will fit in with your existing infrastructure, whether you're on Windows, Mac or Linux.

The first step to having a Harmony Server network is to set up the database server. The database server contains the structure of the database and runs a programs that allows client workstations to open and edit scenes remotely.

Installing a database server can be done on all platforms supported by Harmony: Windows, macOS and GNU/Linux. Although the installation procedure is substantially different on each platform, they all have the same basic components:

- Creating a database file structure and a scene data storage space.
- Sharing the database files using the Samba (Windows) or NFS (macOS or GNU/Linux) file-sharing protocol, so that other machines can access them.
- Configuring the database server program, which handles client requests, to run on startup or login.

Because of the differences in the way Windows and UNIX filesystems work, and in the way Windows, macOS and GNU/Linux support filesharing, extra steps are required if you need to allow Windows clients to connect to a macOS or GNU/Linux-based server, mainly:

- Sharing the database files using the Samba file-sharing protocol.
- Configuring the link server program, which handles the creation of symbolic links for Windows clients.

Additionally, it may be required to add security exceptions to the server machine's firewall for the database server, the link server and the file sharing services used by the server.

Optionally, a server machine's firewall can be fully disabled to allow open communications between different machines on the site. However, a dedicated firewall should be set-up at the site level to ensure computers outside of the site cannot attack machines on the site.



IMPORTANT

Although it may be possible to use macOS or GNU/Linux based client workstations to connect to a Windows based database server, this is not a supported usage scenario and attempting this may cause unexpected problems.

Database Server Installation on Windows

On Windows, the basic setup for a database server is done through the Configuration Wizard, which will create the base structure of the database and automatically configure the database server service.

Disabling the Firewall for a Harmony Database Server on Windows	51
Setting up the Database Server on Windows	52
Removing the Scene Data Storage from the Database Server on Windows	54
Running the Database Server Service on Log-In on Windows	55
Opening the Firewall Ports for a Harmony Database Server on Windows	57
Creating a Scene Data Storage Space on Windows	59
Configuring a Windows Database Server to have a Static IP Address	65
Looking Up the Database Server Log on Windows	69
Launching the Database Server Manually on Windows	70
How to Stop and Restart the Database Server on Windows	71
Pausing and Resuming the Database Server on Windows	73
Uninstalling the Database Server on Windows	74

Disabling the Firewall for a Harmony Database Server on Windows

By default, the Windows firewall will block incoming connections to a database server. This installation guide provides instructions on how to add exceptions for the database server to your firewall. However, you may prefer to disable the firewall completely so as to simplify the installation process or to rule out the firewall as a possible cause when troubleshooting an issue.



IMPORTANT

If you choose to disable the firewall on Windows, it is critical to ensure that your domain has effective site-wide security measures to prevent unauthorized access to your database server.


How to disable the firewall on Windows

1. Open the Windows Control Panel:
 - On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
 - On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.
2. In the Control Panel, select **System and Security**.
3. Under **Windows Defender Firewall**, select **Check firewall status**.
4. In the list to the left, select **Turn Windows Defender Firewall on or off**.
5. Under **Domain network settings**, select **Turn off Windows Defender Firewall (not recommended)**.
6. Under **Private network settings**, select **Turn off Windows Defender Firewall (not recommended)**.
7. Under **Public network settings**, select **Turn off Windows Defender Firewall (not recommended)**.
8. Click on **OK**.

Setting up the Database Server on Windows

Using the Toon Boom Harmony Configuration Wizard, you can quickly create the basic structure for a Harmony database and install the database server as a Windows service, which will run as user **usabatch**.

How to set up a Harmony database server on Windows

1. In the **Start** menu, open the **Programs** menu and select **Harmony 17 [Edition] > Tools >  Configuration Wizard**.
2. When prompted, enable Configuration Wizard to make changes to your device.
The Toon Boom Harmony Configuration Wizard launches.
3. In the list of tasks, right of **Database**, click on the ... button.
4. In the Toon Boom Harmony Database Creation dialog, select **Server/Standalone** and click on **Next**.
5. Under **What drive should be used for the database?**, select which drive on this machine should be used to store the database's files.



NOTE

A **USA_DB** directory will be created at the root of the selected drive.

6. Under **What is the UserName?**, type in the username of the user account you want to create for the database server service. This will create a local user account on your Windows machine, and this user will be connecting to the network and running the service. By default, this use is named **usabatch**.



TIP

If you already configured a user for a database server or the batch processing service on this machine before, you can type in the name of that existing user. The Configuration Wizard will use this existing user account to run the database server service instead of creating a new one.

7. Under **What is the User Password?**, type in the password for the user that will run the database server service. If you are creating a new user, use this field to select their password. If you typed in the user name of an existing user account, type in their actual password.
8. Click on **Next**.

The Scene Data Location dialog appears. In this dialog, you can create a scene data storage on this machine. By default, this dialog is already configured to add a single scene data storage to this machine, on the drive **C**, named **usadata000**.

9. In the User Name and Password fields, make sure the credentials for the scene data location that will be created on your machine match the ones entered for the database server, typed in steps 5 and 6.
10. Click on **Next**.
11. In the Database changes field, verify that that the changes the wizard is about to make are correct, then click on **Finish**.

Your database is set-up. At this point, you should already be able to run Control Center from this machine, log-in with the default user **usabatch** and create scenes and users.

Removing the Scene Data Storage from the Database Server on Windows

By default, Configuration Wizard will add a scene data storage space to the server when creating a database server. If you want all of the scene data storage spaces to be on other machines than the database server machine, you must first remove that storage space from the server, then create a scene data storage space on another machine.

How to remove the scene data storage space from a Windows database server

1. Open an Explorer window.
2. Navigate to the root of the drive in which the scene data storage space was created. By default, this drive is **C:**.
3. Select the directory **usadata000**.
4. Delete this directory.
5. Open the Start menu and, in the Programs list, select **Windows Accessories**.
6. Right-click on **Notepad** and select **Run as Administrator**.
7. In the top menu of Notepad, select **File > Open**.
8. In the Open dialog, select **All Files (*.*)** in the file type drop-down in the bottom-right corner.
9. Browse to and select the following file:

```
C:\Program Files (x86)\Toon Boom Animation\Toon Boom Harmony 17
[Edition]\etc\server.ini
```

In the file, there are numerous lines starting with **FileSystem**. The first one of them will most likely contain information about the scene data storage space the Configuration Wizard created. For example, if the name of the server is **harmonyserver**, this line would contain:

```
FileSystem0=C usadata000 harmonyserver
```

10. On this line, remove all of the information after the equal sign for **FileSystem0**:


```
FileSystem0=
```
11. In the top menu, select **File > Save**.

The storage space has now been removed from your database server. To create another storage space on a different machine, see [Creating a Scene Data Storage Space on Windows](#).

Running the Database Server Service on Log-In on Windows

When creating a database server with Configuration Wizard, the database service is configured to launch as an actual Windows service. This means it launches and allows clients to connect as soon as the machine is booted, without any user being signed-in.

Using Service Launcher, it is possible to configure your database server to only launch its database server service as the logged-in user, when they log-in. This means the database will only be accessible if the user configured to run it is logged in.

Although this may seem impractical, it may be preferable for security reasons. Services installed with the Configuration Wizard are run under the user **usabatch**, an user with administrator privileges which is created by Configuration Wizard to launch the services. When using Service Launcher, the logged-in user does not need to have administrator privileges to run services.


Before configuring Service Launcher to run the database server, the windows service that launches it must be disabled, so as to avoid launching two instances of the database server.



NOTE

While Windows services run regardless of who is logged in and even when no user is logged-in, Service Launcher will only run when the user for which it was specifically configured is logged in. This means that if you want Service Launcher to launch when any user is logged in, you must configure it for each user that has access to the machine.

How to disable the database server service

1. Open the **Start** menu and, in the **Programs** menu, select **Harmony 17 [Edition] > Tools >  Control Panel**.
2. When prompted, allow Control Panel to make changes to your device.
3. In the **Process Starter** drop-down, select **HarmonyStarter0-tbdbserver**.



NOTE

The number after **HarmonyStarter** may be different, depending on how many Harmony services were installed before this one.

4. Click on **Stop**.
5. Click on **Remove**.

From that point on, the database server service will no longer launch on boot.

6. Close the **Control Panel**.



How to configure the database server to launch on user log-in

1. Make sure you are currently signed-in as the user who will run the service.



NOTE

This user will need to be logged in for the service to be running. The user does not need to have administrator privileges.

2. Do one of the following:
 - If Service Launcher is not running, open the **Start** menu and, in the **Programs** list, select **Harmony 17 [Edition] > Tools >  Service Launcher**.
 - If Service Launcher is already running, click on its icon  in the notification tray.

The Service Launcher application launches.

3. In the **Application parameters** section of the **Preferences** tab, make sure that the **Launch on login** option is checked, so that services configured in Service Launcher automatically launch when you log in to your account.
4. Select the **Database Server** tab.
5. Click on **Launch** to verify that the database server launches successfully through Service Launcher.
6. Check the **Launch when Service Launcher starts** option.
7. In the top menu, select **Server > Save All Settings**.

From now on, the database server will launch through Service Launcher when you log in.

Opening the Firewall Ports for a Harmony Database Server on Windows

By default, the Windows firewall will block the communication between client workstations attempting to connect to the Harmony server and the database server service. In order to allow clients to connect to your server, you must open the ports used by the database server on the server machine.

How to open the ports for the Harmony database server on Windows

1. Open the Windows Control Panel:
 - On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
 - On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.

2. Click on **System and Security**.
3. Click on **Windows Defender Firewall**.
4. Click on **Advanced settings**.

The Windows Defender Firewall with Advanced Security window opens.

5. In the list to the left, select **Inbound Rules**.
6. In the **Actions** list to the right, select **New Rule**.

The New Inbound Rule Wizard appears.

7. Under **What type of rule would you like to create?**, select **Port** and click on **Next**.
8. Under **Does this rule apply to TCP or UDP?**, select **TCP**.
9. Under **Does this rule apply to all local ports or specific local ports?**, select **Specify local ports**.
10. In the text field, type the port number **5680**.



NOTE

This is the default port used by the Harmony database server. However, the port number for the database server can be customized by editing the **Dbserver.conf** file located in **/USA_DB**.

11. Click on **Next**.
12. Under **What action should be taken [...]**, select **Allow the connection** and click on **Next**.

13. Under **When does this rule apply?**, check the network types over which clients will attempt to connect to this server, then click on **Next**.
14. In the name field, name the rule "Toon Boom Database Server".
15. Click on **Finish**.

Clients on your network will now be able to communicate with the database server.

Creating a Scene Data Storage Space on Windows

You can create additional storage spaces for your Harmony database on different Windows machines. The database server will connect to these machines to transmit scene data to the clients, allowing you to expand the storage capacity of your database.

The first step is to create a shared folder on the Windows machine that will host the new storage space. Then, you must configure your database server with the information about this new storage space so that, when using Configuration Wizard to set up new Harmony Server clients, this new share will be added to the client configuration. Lastly, if you want the database server itself to have access to the new storage space, you must add a shortcut to that storage space's network path on the database server.

If you intend to configure your database server and the scene data storage spaces to require credentials to be accessed, you should configure all the shares to be accessible with the same user names and password, as Harmony clients will only use one set of credentials to connect to the database and the different scene data storage spaces. For example, in an open network environment, it is typical to give access to all the Harmony-related network shares to the user `usabatch`.

How to create the user `usabatch` on a new Windows-based scene data storage server

1. Open the Windows Control Panel:
 - On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
 - On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.
2. In the Control Panel, click on **System and Security**.
3. In the System and Security panel, click on **Administrative Tools**.
4. Double-click on **Computer Management**.
5. In the Computer Management dialog, in the navigation tree to the left, select **Local Users and Groups > Users**.
6. In the Actions panel to the right, select **More Actions > New User**.
7. In the New User dialog, enter the desired credentials for the user that will be used to access the network share. For example:
 - **User name**: `usabatch`
 - **Full name**: `usabatch`
 - **Password**: `usabatch`
 - **Uncheck** the **User must change password at next logon** option.

- Check the **User cannot change password** option.
 - Check the **Password never expires** option.
8. Click on **OK**.

How to create a scene data storage space on Windows

1. Log-in to the machine that will be used as the new scene data storage space using an account with administrator privileges.
2. Open an Explorer window.
3. Browse to the root of the drive where you want the scene data to be stored (ie: **C:**).
4. Create a new directory.
5. Name the directory **usadata**, followed by a padded, zero-based number indicating its index in the list of storage spaces used by your Harmony database. For example, if this is the second storage space you add to your database, name it **usadata001**.
6. Right-click on the folder and select **Properties**.
7. Open the **Sharing** tab.
8. Click on **Advanced Sharing**.
9. In the Advanced Sharing dialog, check the **Share this folder** checkbox.
10. Click on **Permissions**.
11. In the **Group or user names** list, select **Everyone**.
12. Check the checkbox right of **Full Control**, under the **Allow** column.
13. Click on **OK**.
14. In the Advanced Sharing dialog, click on **OK**.
15. In the Properties dialog, click on **Close**.

How to add the a new scene data storage space to your server configuration

1. Log in to the database server.
2. Open the Start menu and, in the Programs list, select **Windows Accessories**.
3. Right-click on **Notepad** and select *Run as Administrator*.
4. In the top menu of Notepad, select **File > Open**.

5. In the Open dialog, select **All Files (*.*)** in the file type drop-down in the bottom-right corner.
6. Browse to and select the following file:

**C:\Program Files (x86)\Toon Boom Animation\Toon Boom Harmony 17
[Edition]\etc\server.ini**

In the file, there are numerous lines starting with **FileSystem** followed by a number, an equal sign, and some of them will have information after the equal sign. This is the list of scene data storage spaces used by your Harmony server. For example, a newly created database with a single storage space will probably have this scene data storage space configuration:

```
[WizardConfig]
ServerName=SERVERNAME
InstallationDrive=C:\Program Files (x86)\Toon Boom Animation\Toon Boom Harmony
17 [Edition]
UsaShare=usa
UsadbDrive=C
UsadbShare=usa_db
FileSystem0=C usadata000 SERVERNAME
FileSystem1=
FileSystem2=
FileSystem3=
FileSystem4=
[...]
```



NOTE

The **server.ini** file generated by Configuration Wizard contains a file system listing going from 0 to 99. The sample file was shortened here to avoid cluttering the documentation.

7. In the first one of the **FileSystem** lines that has nothing after the equal sign, type in the information for your scene data server, in the following format:

```
FileSystem1=<drive letter> <share name> <server name>
```

Replace the tags between <> brackets with the appropriate values. Make sure to remove the <> brackets from the configuration file as well:

- **<drive letter>**: The driver letter of the drive on which the scene data storage space is located, without any colon or backslash. For example, if the scene data storage space was created on drive **C:**, simply type **C**.

- **<share name>**: The name of the share containing the scene data. Usually, this share is either named **USADATA** or **usadata**, and its name may be followed by a zero-padded, zero-based number indicating the index of the scene data storage space. For example, the first scene data storage space could be named **USADATA** or **usadata000**.
- **<server name>**: The host name of the scene data server. If the scene data storage space is hosted on the same machine as the database, then this host name is the same as the host name for the database.

For example, if the new scene data storage space is the second scene data storage space for the server, and is located in **C:\usadata001** of a computer with the hostname **newserver**, the appropriate configuration for this would be as follows:

```
FileSystem1=C usadata000 newserver
```

8. In the top menu, select **File > Save**.

From now on, the next time you run the Configuration Wizard to set up a Harmony database client on Windows, the new scene data storage space will be automatically added to the client configuration. You can use the Configuration Wizard to re-configure existing clients, if needed.

How to connect the database server to a new remote scene data storage space

1. Log in to the database server.
2. Open the Start menu and, in the Programs list, select **Harmony 17 [Edition] > Tools > Control Panel**.
3. When prompted, allow Control Panel to make changes to your device.
4. In the Control Panel, open the **Connections** tab.
5. In the **Sharename** field, type in the network (UNC) path of the new scene data storage space. This location should start with two backslashes followed by the hostname of the server on which this storage space is hosted, followed by a backslash, then the name of the scene data storage space itself, like in the following example:



```
\\newserver\usadata001
```
6. In the **Username** field, type in the name of a user who has access to this network share.
7. In the **Password** field, type in this user's password.
8. Click on **Create Connection**.
9. Click on **Close**.
10. Right-click on the desktop, then select **New > Shortcut**.

The Create Shortcut dialog appears.

11. Under **Type the location of the item:**, type the network (UNC) path of the new scene data storage space, as in step 5.
12. Click on **Next**.
13. Under **Type a name for this shortcut:**, type in the name of the share and nothing else. For example, if the shortcut path is `\\newserver\usadata001`, the name of the shortcut should be **usadata001**.
14. Click on **Finish**.
15. Right-click on the shortcut and select **Cut**.
16. Open an Explorer window.
17. Browse to the root of the drive with the same drive letter as the one on which the new storage space was added on the scene data server (ie: **C:**)
18. Right-click on the files list and select **Paste**.
19. When prompted, grant administrator permission to move the shortcut.

There is now a shortcut pointing to the new scene data storage space on the database server, allowing it to connect to this storage space and create and manage scenes inside it.

How to add the new storage space to the list of storage spaces used by Harmony and Control Center

1. Open the **Start** menu and, in the **Programs** list, select **Toon BoomHarmony 17 [Edition] > Tools >  Configuration Editor**.
2. When prompted, allow Configuration Editor to make changes to your device.
3. Open the **Manager.conf** tab.
4. Scroll down to the bottom. There should be a line starting with **scene_dir_pattern:** followed by the path to the scene data directories used by your server, except for the one you just created. For example, if the server only had a scene data storage named **usadata000**, the line would be as follows:

```
scene_dir_pattern: /usadata000
```

5. After the last scene data storage space listed on the **scene_dir_pattern** line, add a space, followed by a slash and the name of the new scene data storage space. For example, if you are adding a scene data storage space named **usadata001** and located at the root of your storage

device, you should add it at the end of the line, like so:

```
scene_dir_pattern: /usadata000 /usadata001
```



IMPORTANT

Even on Windows, the path to a scene data directory in Manager.conf must:

- Use forward slashes (/) as separators, not backward slashes (\).
- Start with a forward slash, not a drive letter. For example, instead of `C:\usadata001`, type `/usadata001`.
- Not include the `.lnk` extension of the shortcut file. If you are adding `C:\usadata001.lnk`, simply put `/usadata001`.



TIP

You can also add `*` wildcards in the folder names. For example, this will include all scene data directories at the root of your storage device with a name starting with `usadata`:

```
scene_dir_pattern: /usadata* /USADATA*
```

Configuring a Windows Database Server to have a Static IP Address

It is common to configure server machines to have a static IP. This avoids having to rely on DNS resolution to find the server when its IP address changes, and it also allows you to associate the hostname to the server's static IP address on the machine to skip DNS resolution altogether, increasing network performance.

If your machine is currently connected to a DHCP server that dynamically assigns IP addresses, you can configure your machine to permanently use its current IP address. To do this, you can start by making note of the information for your current network connection, then using these parameters for a permanent IP address.

How to obtain the information about your current connection

1. Open Command Prompt and enter the following command:

```
> ipconfig /all
```

This will output information about your current connection:

```
Windows IP Configuration

Host Name . . . . . : <hostname>
Primary Dns Suffix . . . . . : <domain name>
Node Type . . . . . : Peer-Peer
IP Routing Enabled. . . . . : No
WINS Proxy Enabled. . . . . : No
DNS Suffix Search List. . . . . : <domaine names>

Ethernet adapter Ethernet:

Connection-specific DNS Suffix . . : <domain name>
Description . . . . . : <adapter name>
Physical Address. . . . . : <MAC address>
DHCP Enabled. . . . . : Yes
Autoconfiguration Enabled . . . . : Yes
Link-local IPv6 Address . . . . . : <IPv6 address>
IPv4 Address. . . . . : <IPv4 address>
Subnet Mask . . . . . : <subnet mask>
Lease Obtained. . . . . : <date of lease>
Lease Expires . . . . . : <date of expiration>
Default Gateway . . . . . : <default gateway>
DHCP Server . . . . . : <DHCP server address>
DHCPv6 IAID . . . . . : <DHCPv6 IAID>
DHCPv6 Client DUID. . . . . : <DHCPv6 Client DUID>
DNS Servers . . . . . : <DNS server addresses>
Primary WINS Server . . . . . : <WINS server address>
NetBIOS over Tcpip. . . . . : Enabled
```

2. In the output, under Ethernet adapter Ethernet, make note of the following data:

- The domains in the **DNS Suffix Search List**
- The **IPv4 Address**
- The **Subnet Mask**
- The **Default Gateway** address
- The addresses for the **DNS Servers**

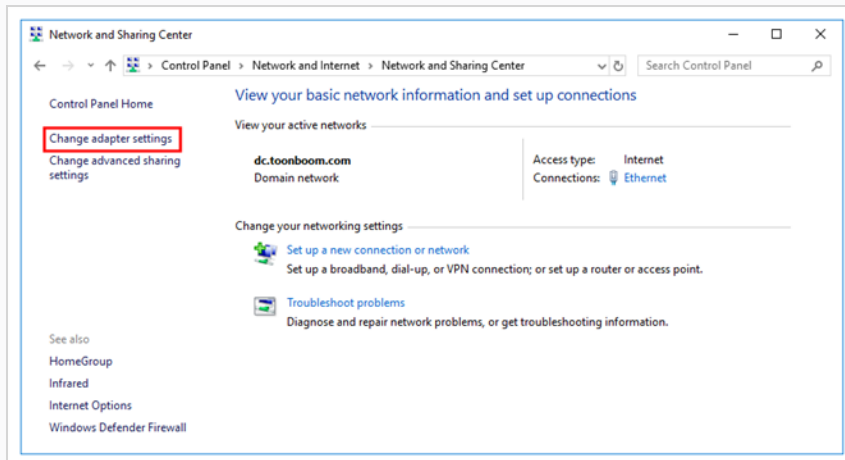
How to configure a Windows server to use a static IP

1. Open the Windows Control Panel:

- On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
- On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.

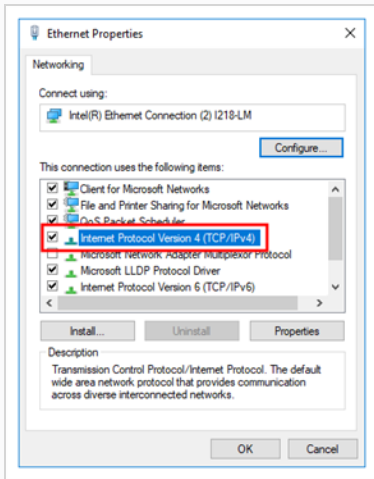
2. Click **View network status and tasks**.

3. On the top-left corner of the Network and Sharing Center, select **Change adapter settings**.



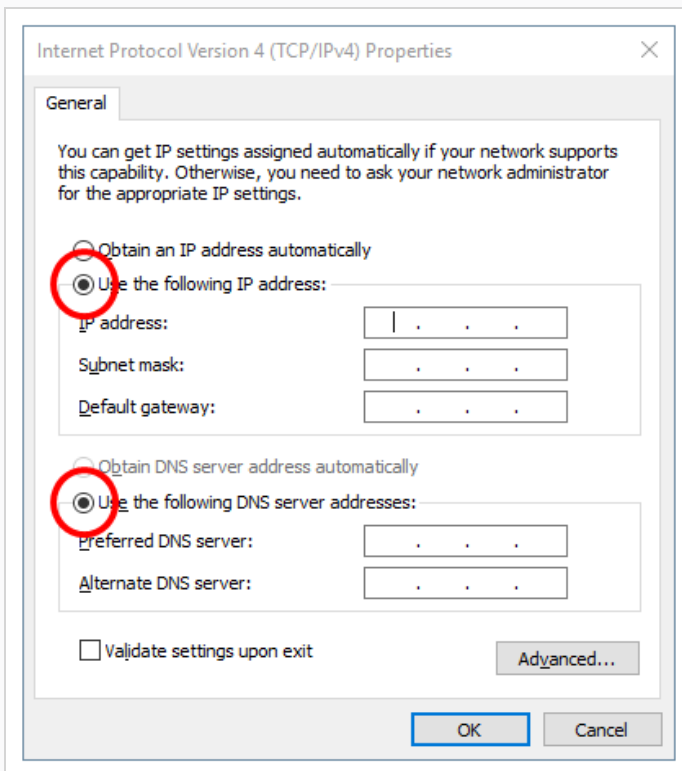
4. Right-click on the network adapter used by your machine to connect to your local network, then select **Properties**.

The Local Area Connection Properties dialog appears.



5. In the items list, select **Internet Protocol Version 4 (TCP/IPv4)**.
6. Click on the **Properties** button under the list.

The Internet Protocol Version 4 (TCP/IPv4) Properties dialog box opens.

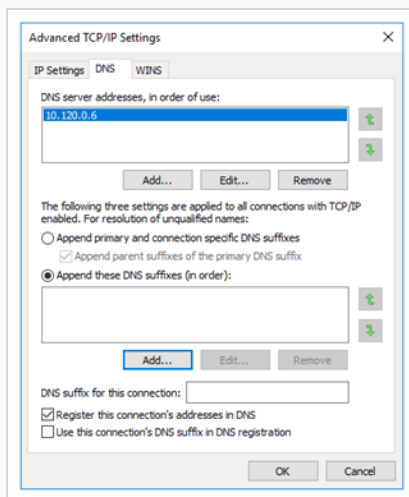


7. Select the **Use the following IP address** option.
8. In the **IP Address** field, enter the **IPv4 Address** you obtained in earlier.

9. In the **Subnet mask** field, enter the **Subnet mask** you obtained earlier.
10. In the **Default gateway** field, enter the **Default gateway** address you obtained earlier.
11. Select the **Use the following DNS server addresses** option.
12. In the **Preferred DNS server** field, enter the **DNS Servers** address obtained earlier.
13. If a second DNS server was listed by ipconfig, type it in the **Alternate DNS server** field. Otherwise, leave this field empty.
14. Click on **Advanced**.

The Advanced TCP/IP Settings dialog opens.

15. Select the **DNS** tab.



16. Select the **Append these DNS suffixes (in order):** option.
17. For each DNS Suffix obtained earlier, click on **Add**, then type in the suffix. Add the suffixes in the order in which there were listed.
18. Click on **OK**.
19. Click on **OK** in the Internet Protocol Version 4 (TCP/IPv4) Properties dialog.
20. Click on **OK** in the Ethernet Properties dialog.


Looking Up the Database Server Log on Windows

In order to troubleshoot problems with the Harmony database server, you can look through its log. This is done differently depending on whether you installed the database server as a Windows service using Configuration Wizard or using Service Launcher.

How to check the log for the database server service

1. Open an Explorer window.
2. Browse to the following location:
`C:\ProgramData\Toon Boom Animation\Toon Boom Harmony`
3. Right-click on `tbdserver.log` and select **Open with > Notepad**.

How to check the log for the database server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the notification tray, click on the  Service Launcher icon.
The Service Launcher application opens.
3. Select the **Database Server** tab.
4. In the bottom half, select the **Log** tab.
5. Click on the **View Complete Log** button in the bottom-right corner.

Launching the Database Server Manually on Windows

You can start the database server directly from a command line. This is usually not practical for a production environment, but it can be very useful when troubleshooting problems. You can view the output of the server directly in the command line as it launches and handles requests.

How to start the database server manually on Windows


1. In a command line, enter the following command:

```
> C:\Program Files (x86)\Toon Boom Animation\Toon Boom Harmony 17  
[Edition]\win64\bin\tbdbserver.exe
```

How to Stop and Restart the Database Server on Windows

If you installed the database server as a Windows service using Configuration Wizard, you can stop it and restart it using the Harmony Control Panel or the Windows Services control panel. If you installed it using Service Launcher, you can stop it and restart it manually in Service Launcher.

How to stop and restart the database server service using the Harmony Control Panel

1. Open the **Start** menu and, in the **Programs** menu, select **Harmony 17 [Edition] > Tools >  Control Panel**.
2. When prompted, allow Control Panel to make changes to your device.
3. In the **Process Starter** drop-down, select **HarmonyStarter0-tbdbserver**.



NOTE

The number after **HarmonyStarter** may be different, depending on how many Harmony services were installed before this one.

4. Click on **Stop**.
5. When you are ready to start the server again, click on **Start**.

How to stop and restart the database server service using the Windows Services control panel

1. Open the Windows Control Panel:
 - On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
 - On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.
2. In the Control Panel, navigate to **System and Security > Administrative Tools**.
3. Double-click on **Services**.


4. Scroll-down to **HarmonyStarter0-tbdbserver**.

**NOTE**

The number after **HarmonyStarter** may be different, depending on how many Harmony services were installed before this one.

5. Right-click on the service and select **Stop**.
6. When you are ready to start the server again, right-click on it and select **Start**.

How to stop and restart the database server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the notification tray, click on the  Service Launcher icon.
The Service Launcher application opens.
3. Select the **Database Server** tab.
4. Click on **Stop**.
5. When you are ready to start the server again, click on **Launch**.

Pausing and Resuming the Database Server on Windows

If you need to perform maintenance or updates on a Harmony server, you can pause it instead of stopping it completely.

Users who attempt to save a scene while the server is paused will receive a warning message. The server will queue the save and commit the changes when the server is resumed.

How to pause the database server

1. In a command prompt, enter the following command:

```
> C:\Program Files (x86)\Toon Boom Animation\Toon Boom Harmony 17  
[Edition]\win64\bin\dbu -p
```

How to check pause status of the database server

1. In a command prompt, enter the following command:

```
> C:\Program Files (x86)\Toon Boom Animation\Toon Boom Harmony 17  
[Edition]\win64\bin\dbu -P
```

How to resume the database server


1. In a command prompt, enter the following command:

```
> C:\Program Files (x86)\Toon Boom Animation\Toon Boom Harmony 17  
[Edition]\win64\bin\dbu -res
```

Uninstalling the Database Server on Windows

If you installed the database server as a Windows service using Configuration Wizard, you can uninstall it using the Harmony Control Panel or the Windows Services control panel. If you installed it using Service Launcher, you can uninstall it in Service Launcher.

How to uninstall the database server service using the Harmony Control Panel

1. Open the **Start** menu and, in the **Programs** menu, select **Harmony 17 [Edition] > Tools >  Control Panel**.
2. When prompted, allow Control Panel to make changes to your device.
3. In the **Process Starter** drop-down, select **HarmonyStarter0-tbdbserver**.



NOTE

The number after **HarmonyStarter** may be different, depending on how many Harmony services were installed before this one.

4. Click on **Stop**.
5. Click on **Remove**.
The database server service will stop and will no longer launch on boot.
6. Close the **Control Panel**.

How to uninstall the database server service using the Windows Services control panel

1. Open the Windows Control Panel:
 - On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
 - On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.
2. In the Control Panel, navigate to **System and Security > Administrative Tools**.
3. Double-click on **Services**.

4. Scroll-down to **HarmonyStarter0-tbdbserver**.




NOTE

The number after **HarmonyStarter** may be different, depending on how many Harmony services were installed before this one.

5. Double-click on the service to open its Properties dialog.
6. In the **Startup type** drop-down, select **Disabled**.
7. Click on **OK**.

From that point on, the database server service will no longer launch on boot.

How to uninstall the database server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the notification tray, click on the  Service Launcher icon.
The Service Launcher application opens.
3. Select the **Database Server** tab.
4. Click on **Stop**.
5. Uncheck the **Launch when Service Launcher starts** option.
6. In the top menu, select **Server > Save All Settings**.

Database Server Installation on macOS

On macOS, you can create a database server by using the Configuration Assistant. The creation of the server is done in two separate steps: The creation of the server's file structure, and the installation of the database server daemon.

Setting the Server Hostname on macOS

By default, unless your macOS server is configured to work with a domain name, macOS will append the suffix `.local` to its chosen hostname. The configuration tools for Harmony will include this suffix when registering the machine's hostname in its configuration.

This can cause a problem when working in a mixed platform environment, as only machines running on macOS can communicate with your machine using the hostname with the `.local` suffix. Machines running Windows or GNU/Linux will only be able to communicate with your macOS machine using its chosen hostname, without the `.local` suffix.

To remedy this, you can manually override the machine's hostname and remove its suffix. This way, the configuration tools for Harmony and the machines trying to communicate with your macOS machine will use the hostname without a suffix.



NOTE

You should only follow this procedure if you do not intend to configure the machines in your network to work with a domain name, as macOS will not append the suffix `.local` to the hostname of computers connected through a domain.

How to manually override the hostname on macOS and remove its `.local` suffix

1. In a terminal, obtain your current machine's hostname, as understood by macOS, with the following command:

```
$ scutil --get HostName
```

2. If the result is the hostname you chose for your machine followed by the suffix `.local`, override this by typing the following command, where `<hostname>` is the hostname you chose for your machine, without the suffix `.local`:

```
$ sudo scutil --set HostName <hostname>
```

3. Restart your machine.

Creating a Harmony Database on macOS

The first step to creating a Harmony database server is to create the database's file structure on the machine that will be used as the server.

How to create a Harmony database on macOS

1. In Finder, proceed to **Applications >  Toon Boom Harmony 17 [Edition]** and double-click **Harmony [Edition]**.

The main Harmony application launches.

2. In the top menu, select **Harmony [Edition] > Tools > Configuration Assistant**.
3. In the Welcome screen, check the **Create a new Toon Boom Harmony database** and uncheck all other options.
4. In the Create Harmony Database dialog, you can configure the options for your database server. It is recommended to use the default options.



IMPORTANT

If the Hostname field displays a value that ends with `.local`, it is highly recommended to remove this suffix before proceeding—see [Setting the Server Hostname on macOS on page 77](#).

5. If you want the database server to also host scene data, leave the **Create an empty Harmony File System (/USADATA)** option checked. If you want to only host scene data on other machines, uncheck this option.
6. Click on Create.
7. When prompted, enter the username and the password of a user with administrator privileges.

The database's base file structure is now created. If you have opted to not create a scene data storage space, see [Creating a Scene Data Storage Server on macOS on page 95](#) to create a storage space for the scenes on your Harmony database.

Installing the Database Server Daemon on macOS

Once you have created a database file structure, you must also enable the database server daemon to allow Harmony to connect to the database. This daemon runs through the user `usabatch`, so this user must be created prior to installing the daemon.



NOTE

The following steps indicate how to make the database server daemon launch on boot, via the user `usabatch`, who will have administrator privileges. If you prefer, you can make the database server launch on log-in as the logged-in user. For more information, see [Running the Database Server on Log-In on macOS on page 81](#).

How to create the user `usabatch` on macOS

1. Open the Apple menu and select **System Preferences**.
2. Click on **Users and Groups**.
3. In the bottom-left corner of the Users & Groups dialog, click on the lock icon labeled **Click the lock to make changes** to get administrator permissions.
4. Enter the username and password of an account with administrator privileges.
5. Click on the + button at the bottom of the user list.

The user creation panel appears.

6. In the **New Account** drop-down, select **Administrator**.
7. In the **Full Name** and **Account Name** fields, enter `usabatch`.
8. Give the user a password.
9. Click on **Create User**.

How to install the database server daemon on macOS

1. In Finder, proceed to **Applications >  Toon Boom Harmony 17 [Edition]** and double-click **Harmony [Edition]**.

The main Harmony application launches.

2. In the top menu, select **Harmony [Edition] > Tools > Configuration Assistant**.
 1. In the Welcome screen, check the **Create the Startup Items** option and uncheck all other options.
 2. Click on Continue.

3. Check the **Create Database Server Startup Item** option and uncheck all other options.
4. Click on **Create**.
5. When prompted, enter the username and the password of a user with administrator privileges.

The database server daemon is now installed and will start running. At this point, you should already be able to run Control Center from this machine, log-in with the default user **usabatch** and create scenes and users.

Running the Database Server on Log-In on macOS

The traditional way of configuring the database server is by using the Configuration Assistant to install it as a daemon. This makes the database server launch immediately on boot, as user **usabatch**.

Using Service Launcher, it is possible to configure your database server to only launch its database server service as the logged-in user. This means the database will only be accessible if the user configured to run it is signed in. Although this may seem impractical, it may be preferable for security reasons, mainly that the logged-in user does not need to have administrator privileges to run services through Service Launcher.

If you have already installed the database server daemon using Configuration Assistant, you will need to disable the daemon first to avoid launching two instances of the database server.



NOTE

While daemons run regardless of who is logged in and even when no user is logged-in, Service Launcher will only run when the user for which it was specifically configured is logged in. This means that if you want Service Launcher to launch when any user is logged in, you must configure it for each user that has access to the machine.

How to disable the database server daemon

1. In a terminal, type in the following command:

```
$ sudo launchctl unload -w  
/Library/LaunchDaemons/com.toonboom.DatabaseServer.plist
```

The database server daemon will stop and will no longer launch on boot.

How to configure the database server to launch on user log-in


1. Make sure you are currently signed-in as the user who will run the service.




NOTE

This user will need to be logged in for the service to be running. The user does not need to have administrator privileges.

2. Do one of the following:

- If Service Launcher is not running, follow these steps to launch it:
 - a. In Finder, proceed to **Applications >  Toon Boom Harmony 17 [Edition]** and double-click **Harmony [Edition]**.

The main Harmony application launches.
 - b. In the top menu, select **Harmony [Edition] > Tools > Service Launcher**.
- If Service Launcher is already running, click on its icon  in the menu bar.

The Service Launcher application window appears.

3. In the **Application parameters** section of the **Preferences** tab, make sure that the **Launch on login** option is checked, so that services configured in Service Launcher automatically launch when you log in to your account.
4. Select the **Database Server** tab.
5. Click on **Launch** to verify that the database server launches successfully through Service Launcher.
6. Check the **Launch when Service Launcher starts** option.
7. In the top menu, select **Server > Save All Settings**.

From now on, the database server will launch through Service Launcher when you log in.

Sharing the Harmony Database Files Using NFS on macOS

Once you have a database file structure and the Harmony database server daemon, you can access the database from the machine on which it was created. To allow other client machines on your network to access the database, you must share its folders using NFS.



NOTE

This procedure will only allow macOS and GNU/Linux based clients to access the database. If you want Windows clients to be able to connect to your Harmony server, see [About Supporting Windows Clients for macOS Servers](#) on page 85.

How to share the database files using the NFS protocol for macOS or GNU/Linux clients

1. In a terminal, enter the following command:

```
$ sudo vi /etc/exports
```

2. Add the following line to allow clients to access the database files:

```
/USA_DB -maproot=nobody
```

3. If your database server also has a scene data storage space, add a line with a similar syntax to share it. For example, if the scene data storage space is named **/USADATA**, add the following line:

```
/USADATA -maproot=nobody
```

4. Save and exit.
5. Verify that the NFS exports are properly configured with the following command:

```
$ sudo nfsd checkexports
```

The command will not return anything if the exports file is properly configured.

6. Make sure the NFS sharing service is enabled with the following command:

```
$ sudo nfsd enable
```

7. If the service was already started, terminate it to force it to load your configuration changes.

```
$ sudo kill -1 `cat /var/run/mountd.pid`
```



NOTE

The back quote (`) character is located at the top-left of the macOS keyboard on the same key as the tilde (~) character. Or simply reboot the computer.

8. Verify that the folders are shared:

```
$ showmount -e
```

The command should output a line for each directory you are sharing using the NFS protocol. For example:

```
/USA_DB Everyone  
/USADATA Everyone
```

You should now be able to mount the shared folders from any macOS or GNU/Linux-based machine on your local network.

About Supporting Windows Clients for macOS Servers

A macOS server can natively handle requests from macOS and GNU/Linux clients. However, in order to allow Windows based clients to connect to the server, extra steps must be followed:

- The Link Server daemon must be installed. This allows Windows clients to create symbolic links on your macOS server.
- Your database files must be shared via the Samba protocol, which is used by Windows clients to access the database.
- The server.ini file must be filled with information on your database server and its scene data storage spaces. This allows Configuration Wizard to configure Windows clients automatically.

Installing the Link Server Daemon on macOS	86
Running the Link Server on Log-In on macOS	87
Sharing the Harmony Database Files Using Samba on macOS	89
Configuring a macOS Database Server for Windows Clients	93

Installing the Link Server Daemon on macOS

If you installed the database server daemon using Configuration Assistant and want your macOS based server to support Windows clients, you must also install the link server daemon.

How to install the database server daemon on macOS

1. In Finder, proceed to **Applications >  Toon Boom Harmony 17 [Edition]** and double-click **Harmony [Edition]**.

The main Harmony application launches.

2. In the top menu, select **Harmony [Edition] > Tools > Configuration Assistant**.
3. In the Welcome screen, check the **Create the Startup Items** option and uncheck all other options.
4. Click on **Continue**.
5. Check the **Create Link Server Startup Item** option and uncheck all other options.
6. Click on **Create**.
7. When prompted, enter the username and the password of a user with administrator privileges.

The link server daemon is now installed and should start immediately.

Running the Link Server on Log-In on macOS

If you configured your database server to launch using Service Launcher, you will also want the Link Server to launch using service launcher, so that it launches in the same circumstances and environment. For more information on running the database server with Service Launcher, see [Running the Database Server on Log-In on macOS on page 81](#).

If you have already installed the link server daemon using Configuration Assistant, you will need to disable the daemon first to avoid launching two instances of the linkserver.



NOTE

While daemons run regardless of who is logged in and even when no user is logged-in, Service Launcher will only run when the user for which it was specifically configured is logged in. This means that if you want Service Launcher to launch when any user is logged in, you must configure it for each user that has access to the machine.

How to disable the link server daemon

1. In a terminal, type in the following command:

```
$ sudo launchctl unload -w  
/Library/LaunchDaemons/com.toonboom.LinkServer.plist
```


How to configure the link server to launch on user log-in

1. Make sure you are currently signed-in as the user who will launch the link server.




NOTE

This user will need to be logged in for the link server to be running. The user does not need to have administrator privileges.

2. Do one of the following:
 - If Service Launcher is not running, follow these steps to launch it:
 - a. In Finder, proceed to **Applications >  Toon Boom Harmony 17 [Edition]** and double-click **Harmony [Edition]**.

The main Harmony application launches.

b. In the top menu, select **Harmony [Edition] > Tools > Service Launcher**.

- If Service Launcher is already running, click on its icon  in the menu bar.

The Service Launcher application window appears.

3. In the **Application parameters** section of the **Preferences** tab, make sure that the **Launch on login** option is checked, so that services configured in Service Launcher automatically launch when you log in to your account.
4. Select the **Link** tab.
5. Click on **Launch** to verify that the Link Server launches successfully through Service Launcher.
6. Check the **Launch when Service Launcher starts** option.
7. In the top menu, select **Server > Save All Settings**.

From now on, the link server will launch through Service Launcher when you log in.

Sharing the Harmony Database Files Using Samba on macOS

Installing Samba for a macOS server is only required if you have Windows based client workstations on your network, as they cannot connect to the database server using NFS. Because Samba is no longer supported by macOS, it must be installed as a third party software on your machine.

Configuring a Harmony database server to share via Samba requires the following steps:

- Disabling the SMB file sharing protocol natively supported by macOS. This protocol may interfere with Samba. Since its options are limited and optimized for personal use, it cannot be used for a Harmony server.
- Installing SMBup, a third-party implementation of Samba for macOS.
- Configuring the shared folders.
- Creating a Sharing type user account. This is required because this implementation of Samba fails to authenticate users that are part of 15 groups or more, and macOS subscribes all users to more than 15 system groups in the back-end, except for Sharing type users.



IMPORTANT

Since this procedure requires installing and configuring a third party software, consider these instructions as indicative only. Using Samba to connect to a macOS based Harmony server cannot be guaranteed to succeed and, even if it succeeds, nothing guarantees it will still work after updating macOS.

How to disable the native SMB file sharing protocol on macOS

1. Open the Apple menu and select **System Preferences**.
2. Click on **Sharing**.
3. In the list to the left, if **File Sharing** is unchecked, then SMB is implicitly disabled. If **File Sharing** is checked, select it.
4. Click on **Options**.
5. Make sure that **Share files and folders using SMB** is unchecked.
6. Click on **Done**.
7. Close the Sharing panel.

How to share the database files using the Samba protocol for Windows clients

1. Download the **SMBUp** package from the following website:

<http://eduo.info/apps/smbup>

2. In Finder, open your **Downloads** directory, then drag and drop the **SMBUp** package into the **Applications** directory.
3. Navigate to the **Applications** directory.
4. Right-click on **SMBUp** and select **Open**.

You will be prompted to confirm that you want to launch the application, as it is from an unidentified developer.

5. Click on **Open**.

After the donation prompt, a message will appear stating that Samba is not installed.

6. Click on **Install Samba for me**.
7. In the next prompt, read the advisory.
8. Click on **Ok, help me install the software**.
9. Click on **Download and install package**.
10. At the prompt, type in the username and password of an account with administrator privileges to allow SMBUp to install Samba.
11. Once the installation is done, click on **Configure Shared Disks**.
12. Near the top-left corner, click on the Lock icon to obtain administrative privileges.
13. Enter the username and password of an account with administrator privileges.
14. In the **Name** field, make sure the name of your Samba server matches the name of your server machine.
15. Click on **Save All**.
16. Close SMBUp. We will configure Samba using the terminal, to make sure it is precisely configured with the right options.
17. In the confirmation prompt, click on **Start Samba for me and Quit Application**.

The Samba service is now installed and enabled.

Configuring Samba shares on macOS

1. In a terminal, open the Samba configuration file to configure Samba and add your database directories as Samba shares:

```
$ vi /opt/local/etc/samba3/smb.conf
```

2. Under the `[global]` section of `smb.conf`, add the following options:

```
[global]
encrypt passwords = yes
blocking locks = no
oplocks = no
level2 oplocks = no
follow symlinks = yes
unix extensions = no
wide links = yes
```

3. At the bottom of the Samba configuration file, add the following:

```
[USA_DB]
comment = Harmony Database
browseable = yes
read only = no
guest ok = no
create mask = 0777
directory mask = 0777
path = /USA_DB
[usa]
comment = Harmony Binaries and Configuration
browseable = yes
read only = no
guest ok = no
create mask = 0777
directory mask = 0777
path = /Applications/Toon Boom Harmony 17 [Edition]/Harmony
[Edition].app/Contents/tba
[USADATA]
comment = Harmony Scene Data 000
browseable = yes
read only = no
guest ok = no
create mask = 0777
directory mask = 0777
path = /USADATA
```

4. Verify that Samba understands the parameters in its configuration file with the following command:

```
$ /opt/local/bin/testparm
```

How to create a user account for Samba on macOS

1. Open the Apple menu and select **System Preferences**.
2. Click on **Users and Groups**.
3. In the bottom-left corner of the Users & Groups dialog, click on the lock icon labeled **Click the lock to make changes** to get administrator permissions.
4. Enter the username and password of an account with administrator privileges.
5. Click on the + button at the bottom of the user list.
The user creation panel appears.
6. In the **New Account** drop-down, select **Sharing Only**.
7. In the **Full Name** and **Account Name** fields, enter the desired name for the user who will have access to the Samba share.
8. Give the user a password.
9. Click on **Create User**.
10. A local user must be added to the list of users with access to the Samba shares. To do this, type in the following command:

```
$ sudo smbpasswd -a usabatch
```

You will be prompted to assign a password to the user, then to confirm it. This username and password will be used to connect to the Samba shares.



NOTES

- The user you are adding to the list of Samba users must already exist as a regular user on the machine.
- On Windows clients, when using the Configuration Wizard to configure Harmony database clients, you must enter this user's username and password in order to be able to connect to the database shares.

Configuring a macOS Database Server for Windows Clients

In order to allow Windows clients to connect to your server using the Configuration Wizard, you must fill in the `server.ini` configuration file, which is located in the `etc` subfolder of the Harmony installation package. This file will provide Configuration Wizard with information on the server as well as of each scene data server it is connected to.

How to configure the `server.ini` file on macOS

1. In a terminal, type in the following command:

```
$ sudo vi "/Applications/Toon Boom Harmony 17 [Edition]/Harmony [Edition].app/Contents/tba/etc/server.ini"
```

2. In `server.ini`, type in the information for the database server as well as the directory name and hostname of every scene data server in your network, like in the following example:

```
[WizardConfig]
ServerName=<server name>
InstallationDrive=C
UsaShare=<harmony share name>
UsadbDrive=C
UsadbShare=<database share name>
```

Replace the tags between `<>` brackets with the appropriate values. Make sure to remove the `<>` brackets from the configuration file as well:

- **<server name>**: The host name of the Harmony database server.
- **<harmony share name>**: The name of the share that gives access to the Harmony binaries and configuration files. By default, this share is named **usa**.
- **<database share name>**: The name of the share that gives access to the database files. By default, this share is named **USA_DB**.

For example, if your database server is called `harmonyserver`, your Harmony installation is shared under the share name **usa** and your Harmony database is shared under the share name **USA_DB**, this would be the proper configuration for this server:

```
[WizardConfig]
ServerName=harmonyserver
InstallationDrive=C
UsaShare=usa
UsadbDrive=C
UsadbShare=USA_DB
```

3. Under the line starting with **UsadbShare**, enter information about each scene data storage space used by your Harmony database, in the following format:

```
FileSystem<file system number>=C <share name> <server name>
```

Replace the tags between <> brackets with the appropriate values. Make sure to remove the <> brackets from the configuration file as well:

- **<file system number>**: The zero-based index of the scene data storage space. For example, the first scene data storage space for your database should be named **FileSystem0** and the second one, **FileSystem1**.
- **<share name>**: The name of the share containing the scene data. Usually, this share is either named **USADATA** or **usadata**, and its name may be followed by a zero-padded, zero-based number indicating the index of the scene data storage space. For example, the first scene data storage space could be named **USADATA**, **usadata000**, **usadata001**, etc.
- **<server name>**: The host name of the scene data server. If the scene data storage space is hosted on the same machine as the database, then this host name is the same as the host name for the database. Otherwise, you must specify the name of the machine on which the scene data storage space is located.

For example, if this is the first scene data storage space for your database, it is hosted on a computer named **dataserver** and its name is **usadata000**, the appropriate configuration for this would be as follows:

```
FileSystem0=C usadata000 dataserver
```



IMPORTANT

The variable names and share names in **server.ini** are case sensitive.



NOTE

server.ini also specifies the drive letter on which the software, the database files and scene data files are stored. Since we are configuring a UNIX-based server, there is no actual drive letter involved. Hence, we are using the drive letter **C** as a placeholder so as to respect the expected format of the **server.ini** configuration file, but the value of these parameters is not relevant and will not affect your configuration.

Creating a Scene Data Storage Server on macOS

You can create additional scene data storage spaces for your Harmony database on machines. The database server will connect to these machines to transmit scene data to the clients, allowing you to expand the storage capacity of your database.

Creating a scene data storage space for your Harmony database requires following these steps:

1. Creating a folder named **usadata** followed by the index number of the storage space at the root of the machine.
2. Sharing the folder via NFS and, if Windows clients must be supported, via Samba as well.
3. Mounting this folder on the database server.
4. If Windows clients must be supported, adding this storage server to the server.ini configuration file.

How to create a scene data storage space on macOS

1. In a terminal, enter the following command, where **###** is the padded, zero-based index of the storage space you are creating, replace **###** with **001**:

```
$ sudo mkdir -m 777 /usadata###
```

A folder in which scene data will be stored is created at the root of the filesystem.

How to share the scene data storage space on macOS

1. Sharing a scene data storage space is done the exact same way as sharing the server's database file system.
 - To share the scene data storage space for macOS and GNU/Linux based clients, see [Sharing the Harmony Database Files Using NFS on macOS on page 83](#).
 - To share the scene data storage space for Windows clients, see [Sharing the Harmony Database Files Using Samba on macOS on page 89](#).

Once this is done, you should already be able to mount the new scene data storage space on the database server as well as on clients.

How to mount the new scene data storage space on a macOS database server

1. First, you should set a permissive file permission mask on the server machine. This will make it so files created by the database server on the scene data server will have the proper permissions to

allow any user to edit them. To do this, see [Setting the User File Creation Permission Mask on macOS on page 169](#).

2. In a terminal, create a folder at the root of the filesystem in which the new scene data storage space will be mounted with the following command, where **<foldername>** is the name of the scene data directory you created on the scene data server:

```
$ sudo mkdir -m 777 /<foldername>
```

3. open the filesystem table in a text editor by typing the following command:

```
$ sudo vi /etc/fstab
```

4. In the filesystem table, add the following line, replacing **<servername>** with the name of the scene data server and **<foldername>** with the name of the scene data directory you created on this scene data server:

```
<servername>: /<foldername> /mnt/<foldername> nfs rw,soft,intr,resvport 0 0
```

5. Save and quit.
6. Attempt to mount the entries of the filesystem table:

```
$ sudo mount -a
```

7. To make sure that it worked, try creating a file in the new scene data storage, and verify that it's visible from both machines. To do this:

- On one machine, enter the following command:

```
$ touch /<foldername>/testfile
```

- On the other machine, enter the following command to verify that **testfile** is present

```
$ ls /<foldername>/
```

How to add the scene data storage file to the server.ini configuration file

1. In a terminal, type in the following command:

```
$ sudo vi "/Applications/Toon Boom Harmony 17 [Edition]/Harmony [Edition].app/Contents/tba/etc/server.ini"
```

If the file was already configured, it most likely contains configuration in the following format:

```
[WizardConfig]
ServerName=<name of Harmony server>
InstallationDrive=C
UsaShare=usa
UsadbDrive=C
```

```
UsadbShare=USA_DB
FileSystem0=C usadata000 <name of first scene data server>
```

2. Under the last line starting with `FileSystem0` which contains information about an existing scene data storage space, enter the information about your new scene data storage space in the following format:

```
FileSystem<file system number>=C <share name> <server name>
```

Replace the tags between `<>` brackets with the appropriate values. Make sure to remove the `<>` brackets from the configuration file as well:

- **<file system number>**: The zero-based index of the scene data storage space. For example, the first scene data storage space for your database should be named **FileSystem0** and the second one, **FileSystem1**.
- **<share name>**: The name of the share containing the scene data. Usually, this share is either named **USADATA** or **usadata**, and its name may be followed by a zero-padded, zero-based number indicating the index of the scene data storage space. For example, the first scene data storage space could be named **USADATA**, **usadata000**, **usadata001**, etc.
- **<server name>**: The host name of the scene data server. If the scene data storage space is hosted on the same machine as the database, then this host name is the same as the host name for the database. Otherwise, you must specify the name of the machine on which the scene data storage space is located.

For example, if this is your second scene data storage space for your database server, it is hosted on a computer named **newdataserver** and is named **usadata001**, its configuration would be as follows:

```
FileSystem1=C usadata001 newdataserver
```



IMPORTANT

The variable names and share names in **server.ini** are case sensitive.

**NOTE**

The first information for a scene data storage space, **C**, is actually the drive letter on which the scene data storage space is located. Since macOS does not use drive letters, there is no drive letter for this storage space, and we are only specifying a drive letter to respect the format of the **server.ini** configuration file. This value does not have an effect your server configuration.

Configuring a macOS Database Server to have a Static IP Address

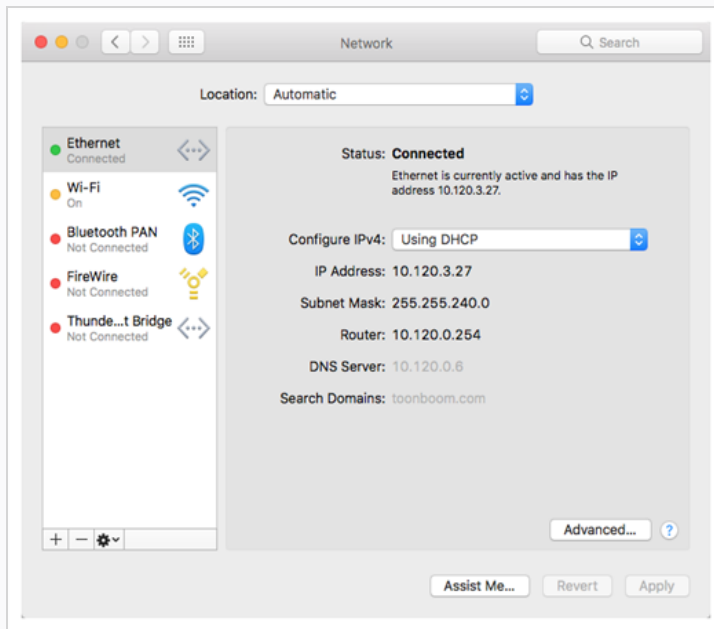
It is common to configure server machines to have a static IP. This avoids having to rely on DNS resolution to find the server when its IP address changes, and it also allows you to associate the hostname to the server's static IP address on the machine to skip DNS resolution altogether, increasing network performance.

If your machine is currently connected to a DHCP server that dynamically assigns IP addresses, you can configure your machine to permanently use its current IP address. To do this, you can start by making note of the information for your current network connection, then using these parameters for a permanent IP address.

How to obtain information about your current connection on macOS

1. Open the Apple menu and select **System Preferences**.
2. Click on **Network**.

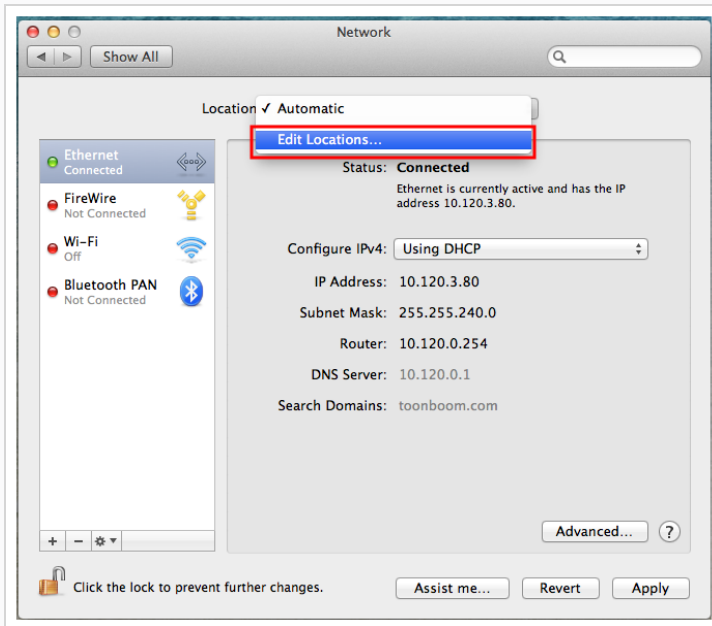
The Network configuration panel appears.



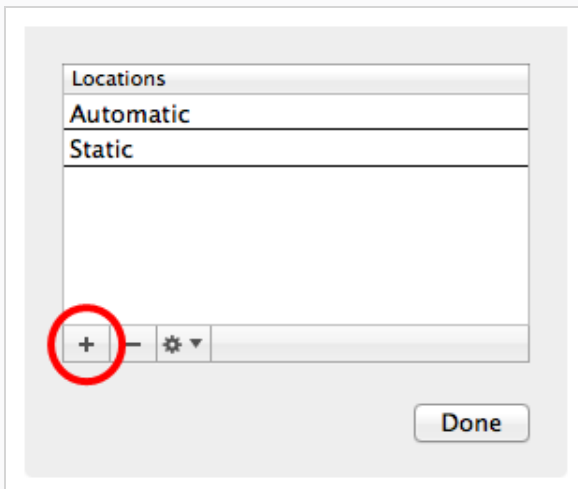
3. In the list to the left, select **Ethernet**.
4. In the panel to the right, make note of the following information:
 - The **IP Address**
 - The **Subnet Mask**
 - The **Router** address
 - The **DNS Server**
 - The **Search Domains**

How to configure a macOS server to use a static IP

1. Open the Apple menu and select **System Preferences**.
2. Click on **Network**.
3. In the list to the left, select **Ethernet**.
4. In the Location drop-down, select Edit Locations.



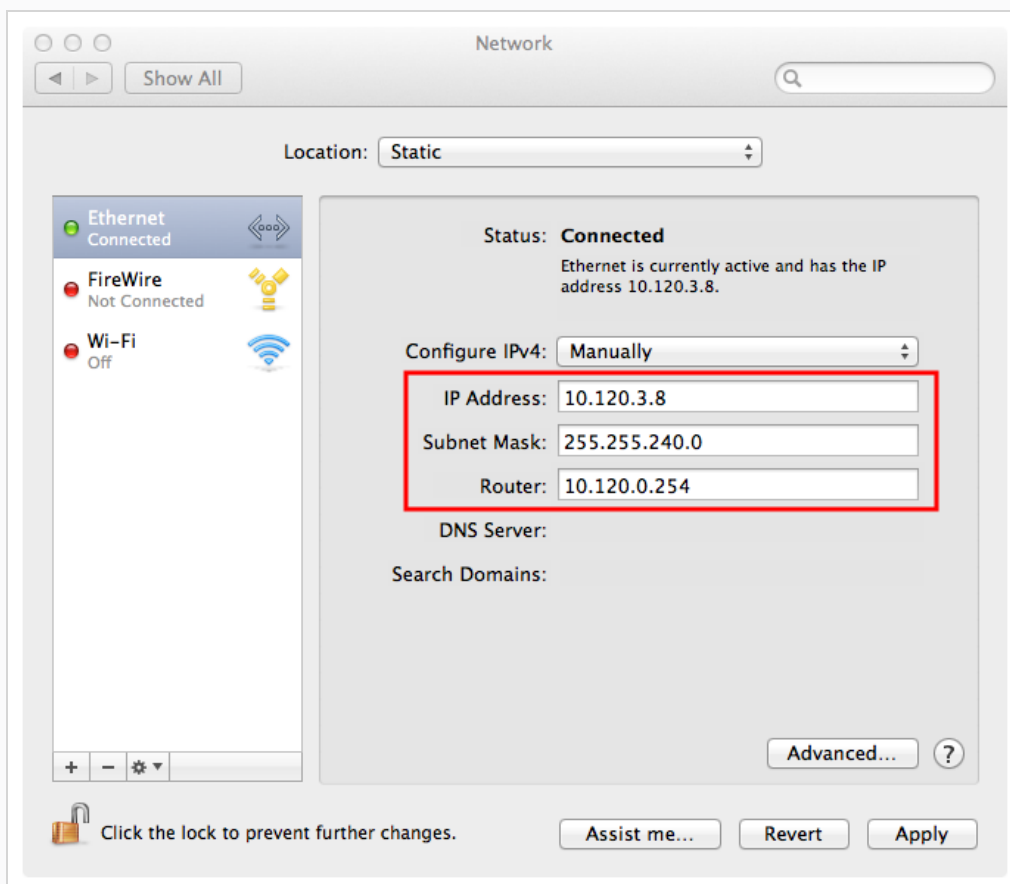
5. Click on the + button to add a location, then name it Static.



6. Click on **Done**.

7. Click on **Advanced**.
8. In the **TCP/IP** tab of the Advanced Network panel, enter the following information:
 - Set the **IPv4 Address** to the **IP Address** obtained earlier.
 - Set the **Subnet Mask** to the **Subnet Mask** obtained earlier.
 - Set the **Router** address to the **Router** address obtained earlier.
9. Open the **DNS** tab.
10. Under the **DNS Servers** list, click on the + button to add a DNS server. Add each of the DNS Server addresses obtained earlier.
11. Under the **Search Domains** list, click on the + button to add a search domain. Add each of the search domains obtained earlier.
12. Click on **OK**.

If your configuration works, the Network panel will indicate **Status: Connected** at the top.



Looking Up the Database Server Log on macOS

In order to troubleshoot problems with the Harmony database server, you can look through its log. This is done differently depending on whether you installed the database server as a daemon using Configuration Assistant, or using Service Launcher.

If your server is configured for Windows clients, you can also check the link server log for issues.

How to check the log for the database server daemon

1. In a terminal, enter the following command:


```
$ less /tmp/com.toonboom.DatabaseServer.log
```

How to check the log for the link server daemon


1. In a terminal, enter the following command:

```
$ less /tmp/com.toonboom.LinkServer.log
```

How to check the log for the database server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the top menu, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **Database Server** tab.
4. In the bottom half, select the **Log** tab.
5. Click on the **View Complete Log** button in the bottom-right corner.

How to check the log for the link server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the top menu, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **Link** tab.
4. In the bottom half, select the **Log** tab.

5. Click on the **View Complete Log** button in the bottom-right corner.

Launching the Database Server Manually in macOS

You can start the database server directly from a terminal. This is usually not practical for a production environment, but it can be very useful when troubleshooting problems.

If your server needs to handle Windows clients, you can also launch the link server in a different terminal to ensure that both services work.

You can view the output of both services directly in the terminal in which you launch them as they start and handle requests.

How to start the database server manually

1. In a terminal, enter the following command:

```
$ "/Applications/Toon Boom Harmony 17 [Edition]/Harmony  
[Edition].app/Contents/tba/macosx/bin/tbdbserver"
```

How to start the link server manually

1. In another terminal, enter the following command:

```
$ "/Applications/Toon Boom Harmony 17 [Edition]/Harmony  
[Edition].app/Contents/tba/macosx/bin/Link_srv"
```

Stopping and Restarting the Database Server on macOS

If you installed the database server as a daemon using the Configuration Assistant, you can stop it and restart it using the `launchctl` command line tool. If you installed it using Service Launcher, you can stop it and restart it manually in Service Launcher.

If you have installed the link server so as to be able to handle Windows clients, stopping and restarting it may also be necessary for the intended purpose.

How to stop and restart the database server daemon

1. In a terminal, enter the following command:

```
$ sudo launchctl unload  
/Library/LaunchDaemons/com.toonboom.DatabaseServer.plist
```

2. When you are ready to start the server again, enter the following command:

```
$ sudo launchctl load  
/Library/LaunchDaemons/com.toonboom.DatabaseServer.plist
```

How to stop and restart the link server daemon


1. In a terminal, enter the following command:

```
$ sudo launchctl unload  
/Library/LaunchDaemons/com.toonboom.LinkServer.plist
```


2. When you are ready to start the server again, enter the following command:

```
$ sudo launchctl load  
/Library/LaunchDaemons/com.toonboom.LinkServer.plist
```

How to stop and restart the database server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the top menu, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **Database Server** tab.
4. Click on **Stop**.
5. When you are ready to start the server again, click on **Launch**.

How to stop and restart the link server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the top menu, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **Link Server** tab.
4. Click on **Stop**.
5. When you are ready to start the server again, click on **Launch**.

Pausing and Resuming the Database Server on macOS

If you need to perform maintenance or updates on a Harmony server, you can pause it instead of stopping it completely.

Users who attempt to save a scene while the server is paused will receive a warning message. The server will queue the save and commit the changes when the server is resumed.

How to pause the database server

1. In a command prompt, enter the following command:

```
$ "/Applications/Toon Boom Harmony 17 [Edition]/Harmony  
[Edition].app/Contents/tba/macosx/bin/dbu" -p
```

How to check pause status of the database server

1. In a command prompt, enter the following command:

```
$ "/Applications/Toon Boom Harmony 17 [Edition]/Harmony  
[Edition].app/Contents/tba/macosx/bin/dbu" -P
```

How to resume the database server

1. In a command prompt, enter the following command:

```
$ "/Applications/Toon Boom Harmony 17 [Edition]/Harmony  
[Edition].app/Contents/tba/macosx/bin/dbu" -res
```

Uninstalling the Database Server on macOS

If you installed the database server as a daemon using the Configuraton Assistant, you can uninstall it using the `launchctl` command line tool. If you installed it using Service Launcher, you can uninstall it in Service Launcher.

How to uninstall the database server daemon

1. In a terminal, enter the following command:

```
$ sudo launchctl unload -w  
/Library/LaunchDaemons/com.toonboom.DatabaseServer.plist
```

The daemon will no longer launch on boot.

2. If you want, you can delete the daemon permanently with the following command:

```
$ sudo rm /Library/LaunchDaemons/com.toonboom.DatabaseServer.plist
```

How to uninstall the link server daemon

1. In a terminal, enter the following command:


```
$ sudo launchctl unload -w  
/Library/LaunchDaemons/com.toonboom.LinkServer.plist
```

The daemon will no longer launch on boot.


2. If you want, you can delete the daemon permanently with the following command:

```
$ sudo rm /Library/LaunchDaemons/com.toonboom.LinkServer.plist
```

How to uninstall the database server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the top menu, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **Database Server** tab.
4. Click on **Stop**.
5. Uncheck the **Launch when Service Launcher starts** option.
6. In the top menu, select **Server > Save All Settings**.

How to uninstall the link server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the top menu, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **Link Server** tab.
4. Click on **Stop**.
5. Uncheck the **Launch when Service Launcher starts** option.
6. In the top menu, select **Server > Save All Settings**.

Database Server Installation on GNU/Linux

On GNU/Linux, you can create a database server with the help of a script that creates the basic database file structure, then enabling the database server daemon which was created during the installation of Harmony.

Disabling Security for a Harmony Database Server on GNU/Linux

GNU/Linux comes bundled with two core security components:

- A firewall (typically either **iptables** or **firewalld**), which will block incoming connections that were not specifically allowed.
- **SELinux**, a kernel security module that enforces access control policies on software, files and network resources.

The default configuration for these components will prevent clients from connecting to a Harmony database server. This installation guide provides instructions on how to add security exceptions to these components so that they do not have to be disabled. However, you may prefer to disable them so as to simplify the installation process or to rule out security components as possible causes when troubleshooting an issue.



IMPORTANT

If you choose to disable the firewall and SELinux, it is critical to ensure that your domain has effective site-wide security measures to prevent unauthorized access to your database server.

How to disable the firewall on CentOS 6, RedHat Enterprise Linux 6 or other systems using the iptables service

1. In a terminal, enter the following commands:

```
$ sudo chkconfig iptables off
$ sudo service iptables stop
```

How to disable the firewall on CentOS 7, RedHat Enterprise Linux 7 or other systems using the firewalld service

1. In a terminal, enter the following commands:

```
$ sudo systemctl disable firewalld
$ sudo systemctl stop firewalld
```

How to disable SELinux

1. In a terminal, enter the following command:

```
$ sudo vi /etc/selinux/config
```

2. Find the line with the following parameter:


```
SELINUX=enforcing
```

3. Replace **enforcing** with **disabled**:

```
SELINUX=disabled
```

**IMPORTANT**

Double-check to make sure you did not make any typo. An error in the SELinux configuration file may cause GNU/Linux to fail to boot.

4. Save and quit.
5. Reboot to apply the new configuration.

Creating a Harmony Database on GNU/Linux

The first step to creating a Harmony database server is to create the database's file structure on the machine that will be used as the server. You can accomplish this by running the `create_usa_db` script from Harmony's `bin` directory.

The basic database file structure is in the `USA_DB` directory. However, this is not where scene data is stored. If you want the database server to be able to host scene data, you can also create an empty scene data storage space alongside your database file structure.



NOTE

The following steps only indicate how to create a database file structure using the `create_usa_db` script. If you prefer, you can create the database file structure and enable the database server daemon simultaneously using the Harmony installation script—see [Installing the Database Server Daemon on GNU/Linux](#) on page 114.

How to create a database file structure using the `create_usa_db` script

1. In a terminal, navigate to the directory in which the Harmony binaries were installed:

```
$ cd /usr/local/ToonBoomAnimation/harmony[Edition]_17/lnx86_64/bin
```

2. Run the database creation script:

```
$ sudo ./create_usa_db
```

The script will create the basic file structure for a Harmony database in `/USA_DB`.

How to create a scene data storage space

1. In a terminal, enter the following command:

```
$ sudo mkdir -m 777 /usadata000
```

A scene data storage directory named `usadata000` is now located at the root of your database server's file system.

Installing the Database Server Daemon on GNU/Linux

Once you have created a database file structure, you must also enable the database server daemon to allow Harmony to connect to the database. This daemon runs through the user `usabatch`, so this user must be created prior to installing the daemon.

The Harmony installer for GNU/Linux always installs daemon scripts on your machine, but does not enable them by default. Hence, there are two approaches to installing the database server daemon:

- By enabling the daemon that was already installed with Harmony using the `chkconfig` command.
- By running the Harmony installation script with the `-d` parameter. This will also create a database file structure on your machine if none is present.

This daemon runs through the user `usabatch`, so this user must be created prior to installing the daemon.



NOTE

The following steps indicate how to make the database server daemon launch on boot, via the user `usabatch`. If you prefer, you can make the database server launch on log-in as the logged-in user. For more information, see [Running the Database Server on Log-In on GNU/Linux on page 116](#).

How to create the `usabatch` user on GNU/Linux

1. In a terminal, enter the following command:

```
$ sudo useradd -m usabatch
```

```
$ sudo passwd usabatch
```

2. When prompted, enter the password you want to assign to the user `usabatch`, then confirm it.

The user `usabatch` is now created and can be used to run Harmony daemons.

How to enable the database server daemon on GNU/Linux

1. In a terminal, enter the following command to enable the database server daemon:

```
$ sudo chkconfig toonboom_dbserver on
```

If the command returns nothing, it succeeded. The database server will launch next time you boot the machine.

2. To launch the database server daemon immediately, run the following command:

```
$ sudo service toonboom_dbserver start
```

The command will report whether it succeeded or failed.

How to create a database file structure and enable the database server daemon simultaneously using the Harmony installation script

1. In a terminal, navigate to the directory where you downloaded the installation package.

```
$ cd ~/Downloads
```

2. Extract the package's contents in the directory.

```
$ tar -xvzf ./name-of-package.tar.gz
```

3. Switch to the directory in which the package's contents were extracted.

```
$ cd name-of-package
```

4. Run the installer with the -d parameter:

```
$ sudo ./install -d
```

5. When prompted, confirm that you want to proceed with the installation.

The installation script will create a database file structure in `/USA_DB`, enable the database server daemon and start it.



NOTE

The installation script will not create a database file structure in `/USA_DB` if this directory already exists, whether it is a local database or a remote database mounted on the file system.

Running the Database Server on Log-In on GNU/Linux

The traditional way of running the Harmony database server is by installing it as a daemon. This makes the database server launch immediately on boot, as user **usabatch**.

Using Service Launcher, it is possible to configure your database server to only launch its database server service as the logged-in user. This means the database will only be accessible if the user configured to run it is signed in. Although this may seem impractical, it may be preferable for security reasons, mainly that the logged-in user does not need to have administrator privileges to run services through Service Launcher.

If you have enabled the database server daemon, you will need to disable and stop it before you can configure it to launch with Service Launcher, to avoid launching two instances of the database server.



NOTE

While daemons run regardless of who is logged in and even when no user is logged-in, Service Launcher will only run when the user for which it was specifically configured is logged in. This means that if you want Service Launcher to launch when any user is logged in, you must configure it for each user that has access to the machine.

How to disable the database server daemon

1. In a terminal, type in the following commands to stop, then disable the daemon:

```
$ sudo service toonboom_dbserver stop
$ sudo chkconfig toonboom_dbserver off
```

The database server daemon will stop and will no longer launch on boot.


How to configure the database server to launch on user log-in


1. Make sure you are currently signed-in as the user who will run the service.



NOTE

This user will need to be logged in for the service to be running. The user does not need to have administrator privileges.

2. Do one of the following:
 - If Service Launcher is not running, in the GNOME top menu, select **Applications > Harmony 17 [Edition] >  Service Launcher**.

- If Service Launcher is already running, click on its icon  in the system tray.

The Service Launcher application starts.

3. Select the **Database Server** tab.
4. Click on **Launch** to verify that the database server launches successfully through Service Launcher.
5. Check the **Launch when Service Launcher starts** option.
6. In the top menu, select **Server > Save All Settings**.

From now on, the database server will launch through Service Launcher when you log in.

Expanding the File Descriptor Limit for a Database Server on GNU/Linux

If your database server is likely to have a lot of connected clients, you should expand the file descriptor limit for the user who will run the database server daemon.

How to expand the file descriptor limit for the database server

1. In a terminal, open the limits configuration file in a text editor:

```
$ sudo vi /etc/security/limits.conf
```

2. At the end of the file, just before the **# End of file** line, add the following lines, replacing **<username>** with the name of the user who runs the database server daemon:

```
<username> soft nofile 16384
<username> soft nofile 16384
```

For example, if you installed the database server as a daemon, it will run as user `usabatch` by default. Hence you would need to add the following lines before **# End of file**:

```
usabatch soft nofile 16384
usabatch soft nofile 16384
```

3. Save and quit.

Opening the Firewall Ports for a Harmony Database Server on GNU/Linux

The GNU/Linux firewall will block attempts from other computers to communicate with the database server. In order to circumvent this, the ports used by the database server must be opened.

How to allow connections to the database server to go through the firewall on CentOS 6, RedHat Enterprise Linux 6 or other systems using the iptables service

1. Add the ports used by the database server to the list of open ports with the following commands:

```
$ sudo iptables -I INPUT -m state --state NEW -p tcp -m tcp --dport 5678 -j ACCEPT
$ sudo iptables -I INPUT -m state --state NEW -p tcp -m tcp --dport 5680 -j ACCEPT
$ sudo service iptables save
$ sudo service iptables restart
```

The GNU/Linux firewall should now allow other machines to connect to the database server.



NOTE

Ports **5678** and **5680** are the default ports used by the Harmony Database server. You can customize which ports the Database server uses by editing the `/USA_DB/Dbserver.conf` configuration file.

How to allow connections to the database server to go through the firewall on CentOS 7, RedHat Enterprise Linux 7 or other systems using the firewalld service

1. Add the ports used by the database server to the list of open ports with the following commands:

```
$ sudo firewall-cmd --permanent --zone=public --add-port=5678/tcp
$ sudo firewall-cmd --permanent --zone=public --add-port=5680/tcp
$ sudo firewall-cmd --reload
```

The GNU/Linux firewall should now allow other machines to connect to the database server.



NOTE

Ports **5678** and **5680** are the default ports used by the Harmony Database server. You can customize which ports the Database server uses by editing the `/USA_DB/Dbserver.conf` configuration file.

Sharing the Harmony Database Files Using NFS on GNU/Linux

Once you have a database file structure and the Harmony database server daemon, you can access the database from the machine on which it was created. To allow other client machines on your network to access the database, you must share its folders using NFS.

If the GNU/Linux firewall is enabled, you must add rules to allow remote connections to the NFS server so as to allow other machines to connect to the NFS shares.



NOTE

This procedure will only allow GNU/Linux and macOS based clients to access the database. If you want Windows clients to be able to connect to your Harmony server, see [About Supporting Windows Clients for macOS Servers](#) on page 85.

How to configure the NFS daemon to share your database folders over the network

1. Make sure the NFS service and its utilities are installed:

```
$ sudo yum install nfs-utils
```

2. Enable and start the rpcbind daemon, the NFS daemon, the NFS lock daemon:

```
$ sudo chkconfig rpcbind on
$ sudo service rpcbind start
$ sudo chkconfig nfs on
$ sudo service nfs start
$ sudo chkconfig nfslock on
$ sudo service nfslock start
```

3. Then, open the configuration file `/etc/exports` in `vi` to add your database's share directories:

```
$ sudo vi /etc/exports
```

1. Add the following line to allow clients to access the database files:

```
/USA_DB *(rw, sync)
```

2. If your database server also has a scene data storage space, add a line with a similar syntax to share it. For example, if the scene data storage space is named `/usadata000`, add the following line:

```
/usadata000 *(rw, sync)
```

3. Save and exit.
4. Run the export command to update the NFS daemon with the new configuration:

```
$ sudo exportfs -r
```

5. Type this command to verify that the new configuration is in effect:

```
$ showmount --exports
```

The utility will list all the directories currently shared through NFS.

How to allow connections to the NFS shares to go through the firewall on CentOS 6, RedHat Enterprise Linux 6 or other systems using the iptables service

1. Open the NFS configuration file in a text editor:

```
$ sudo vi /etc/sysconfig/nfs
```

2. In the NFS configuration file, remove the hash # sign before the lines with the following content, so as to uncomment them. This will make sure the NFS service always uses the same ports, and hence, will allow us to safely open those ports in the firewall:

- RQUOTAD_PORT=875
- LOCKD_TCPPORT=32803
- LOCKD_UDPPORT=32769
- MOUNTD_PORT=892
- STATD_PORT=662
- STATD_OUTGOING_PORT=2020

3. Add the ports and their protocols to the list of open ports with the following commands:

```
$ sudo iptables -I INPUT -m state --state NEW -m tcp -p tcp --dport 2049 -j ACCEPT
$ sudo iptables -I INPUT -m state --state NEW -m tcp -p tcp --dport 111 -j ACCEPT
$ sudo iptables -I INPUT -m state --state NEW -m udp -p udp --dport 111 -j ACCEPT
$ sudo iptables -I INPUT -m state --state NEW -m tcp -p tcp --dport 32803 -j ACCEPT
$ sudo iptables -I INPUT -m state --state NEW -m udp -p udp --dport 32769 -j ACCEPT
$ sudo iptables -I INPUT -m state --state NEW -m tcp -p tcp --dport 892 -j ACCEPT
$ sudo iptables -I INPUT -m state --state NEW -m udp -p udp --dport 892 -j ACCEPT
$ sudo iptables -I INPUT -m state --state NEW -m tcp -p tcp --dport 875 -j ACCEPT
$ sudo iptables -I INPUT -m state --state NEW -m udp -p udp --dport 875 -j ACCEPT
$ sudo iptables -I INPUT -m state --state NEW -m tcp -p tcp --dport 662 -j ACCEPT
```

```
$ sudo iptables -I INPUT -m state --state NEW -m udp -p udp --dport
662 -j ACCEPT
$ sudo service iptables save
$ sudo service iptables restart
```

Other machines should now be able to mount the NFS shares.

How to allow connections to the NFS shares to go through the firewall on CentOS 7, RedHat Enterprise Linux 7 or other systems using the firewalld service

1. In a terminal, enter the following commands:

```
$ firewall-cmd --permanent --add-service=nfs
$ firewall-cmd --permanent --add-service=mountd
$ firewall-cmd --permanent --add-service=rpc-bind
$ firewall-cmd --reload
```

Other machines should now be able to mount the NFS shares.

About Supporting Windows Clients for GNU/Linux Servers

A GNU/Linux server can natively handle requests from macOS and GNU/Linux clients. However, in order to allow Windows based clients to connect to the server, extra steps must be followed:

- The Link Server daemon must be installed. This allows Windows clients to create symbolic links on your macOS server.
- Your database files must be shared via the Samba protocol, which is used by Windows clients to access the database.
- The server.ini file must be filled with information on your database server and its scene data storage spaces. This allows Configuration Wizard to configure Windows clients automatically.

Installing the Link Server Daemon on GNU/Linux	124
Running the Link Server on Log-In on GNU/Linux	126
Opening the Firewall Ports for the Link Server on GNU/Linux	128
Sharing the Harmony Database Files Using Samba on GNU/Linux	129
Opening the Firewall Ports and Configuring SELinux for a Samba Server on GNU/Linux	131
Configuring a GNU/Linux Database Server for Windows Clients	133

Installing the Link Server Daemon on GNU/Linux

If you installed the database server daemon and want your GNU/Linux based server to support Windows clients, you must also install the link server daemon. This daemon runs through the user `usabatch`, so this user must be created prior to installing the daemon.

How to create the `usabatch` user on GNU/Linux

1. In a terminal, enter the following command:

```
$ sudo useradd -m usabatch
$ sudo passwd usabatch
```

2. When prompted, enter the password you want to assign to the user `usabatch`, then confirm it.

The user `usabatch` is now created and can be used to run Harmony daemons.

How to enable the link server daemon on GNU/Linux

1. In a terminal, enter the following command to enable the link server daemon:

```
$ sudo chkconfig toonboom_link_srv on
```

If the command returns nothing, it succeeded. The link server will launch next time you boot the machine.

2. To launch the link server daemon immediately, run the following command:

```
$ sudo service toonboom_link_srv start
```

The command will report whether it succeeded or failed.

How to enable the link server daemon using the Harmony installation script

1. In a terminal, navigate to the directory where you downloaded the installation package.

```
$ cd ~/Downloads
```

2. Extract the package's contents in the directory.

```
$ tar -xvzf ./name-of-package.tar.gz
```

3. Switch to the directory in which the package's contents were extracted.

```
$ cd name-of-package
```

4. Run the installer with the -d parameter:

```
$ sudo ./install -d
```

5. When prompted, confirm that you want to proceed with the installation.

The installation script will enable and start the link server daemon.

Running the Link Server on Log-In on GNU/Linux

If you configured your database server to launch using Service Launcher, you will also want the Link Server to launch using Service Launcher, so that it launches in the same circumstances and environment. For more information on running the database server with Service Launcher, see [Running the Database Server on Log-In on GNU/Linux on page 116](#).

If you have already enabled the link server daemon, you will need to disable the daemon first to avoid launching two instances of the link server.



NOTE

While daemons run regardless of who is logged in and even when no user is logged-in, Service Launcher will only run when the user for which it was specifically configured is logged in. This means that if you want Service Launcher to launch when any user is logged in, you must configure it for each user that has access to the machine.

How to disable the link server daemon

1. In a terminal, type in the following commands to stop, then disable the daemon:

```
$ sudo service toonboom_link_srv stop
$ sudo chkconfig toonboom_link_srv off
```

The link server daemon will stop and will no longer launch on boot.



How to configure the link server to launch on log-in

1. Make sure you are currently signed-in as the user who will run the service.



NOTE

This user will need to be logged in for the service to be running. The user does not need to have administrator privileges.

2. Do one of the following:
 - If Service Launcher is not running, in the GNOME top menu, select **Applications > Harmony 17 [Edition] >  Service Launcher**.
 - If Service Launcher is already running, click on its icon  in the system tray.

The Service Launcher application starts.

3. In the **Application parameters** section of the **Preferences** tab, make sure that the **Launch on login** option is checked, so that services configured in Service Launcher automatically launch when you log in to your account.
4. Select the **Link** tab.
5. Click on **Launch** to verify that the Link Server launches successfully through Service Launcher.
6. Check the **Launch when Service Launcher starts** option.
7. In the top menu, select **Server > Save All Settings**.

From now on, the link server will launch through Service Launcher when you log in.

Opening the Firewall Ports for the Link Server on GNU/Linux

The GNU/Linux firewall will block attempts from other computers to communicate with the database server. In order to circumvent this, the ports used by the database server must be opened.

How to allow connections to the Samba shares to go through the firewall on CentOS 6, RedHat Enterprise Linux 6 or other systems using the iptables service

1. Add the port used by the link server to the list of open ports with the following command:

```
$ sudo iptables -I INPUT -m state --state NEW -p tcp -m tcp --dport 5679 -j ACCEPT
$ sudo service iptables save
$ sudo service iptables restart
```

The GNU/Linux firewall should now allow other machines to connect to the database server.



NOTE

Port **5679** is the default port used by the Harmony Link server. You can customize which port the Link server uses by editing the `/USA_DB/Link_srv.conf` configuration file.

How to allow connections to the Samba shares to go through the firewall on CentOS 7, RedHat Enterprise Linux 7 or other systems using the firewalld service

1. Add the port used by the link server to the list of open ports with the following command:

```
$ sudo firewall-cmd --permanent --zone=public --add-port=5679/tcp
$ sudo firewall-cmd --reload
```

The GNU/Linux firewall should now allow other machines to connect to the database server.



NOTE

Port **5679** is the default port used by the Harmony Link server. You can customize which port the Link server uses by editing the `/USA_DB/Link_srv.conf` configuration file.

Sharing the Harmony Database Files Using Samba on GNU/Linux

In order to allow Windows clients to access the database's directories remotely, they must be shared by the server using the Samba protocol.



NOTE

The following commands require elevated privileges. To execute them successfully, you must either be logged in as a user with root privileges, or precede each command with **sudo**.

How to install and configure Samba to share your database folders over the network

1. In a terminal, type the following command to install Samba:

```
$ sudo yum install samba
```

If samba is not installed, you will be prompted to confirm the installation. Confirm and wait for the installation to finish. If it is already installed, yum will advise you of this and quit.

2. Open the Samba configuration file to configure Samba and add your database directories as Samba shares:

```
$ sudo vi /etc/samba/smb.conf
```

3. Under the **[global]** section of smb.conf, add the following options:

```
[global]
encrypt passwords = yes
blocking locks = no
oplocks = no
level2 oplocks = no
follow symlinks = yes
unix extensions = no
wide links = yes
```

4. At the bottom of the Samba configuration file, add the following:

```
[USA_DB]
comment = Harmony Database
browseable = yes
read only = no
guest ok = no
create mask = 0777
directory mask = 0777
path = /USA_DB
[usa]
comment = Harmony Binaries and Configuration
browseable = yes
read only = no
guest ok = no
```

```
create mask    = 0777
directory mask = 0777
path          = /usr/local/ToonBoomAnimation/harmony[Edition]_17
[usadata000]
comment       = Harmony Scene Data 000
browseable    = yes
read only     = no
guest ok      = no
create mask   = 0777
directory mask = 0777
path         = /usadata000
```

5. Verify that Samba understands the parameters in its configuration file with the following command:

```
$ testparm
```

6. A local user must be added to the list of users with access to the Samba shares. To do this, type in the following command:

```
$ sudo smbpasswd -a usabatch
```

You will be prompted to assign a password to the user, then to confirm it. This username and password will be used to connect to the Samba shares.

**NOTE**

The user you are adding to the list of Samba users must already exist as a regular user on the machine.

7. Enable and start the Samba daemon:

```
$ sudo chkconfig smb on
```

```
$ sudo service smb start
```

Samba is now configured to give access to the database files with the selected username and password.

Opening the Firewall Ports and Configuring SELinux for a Samba Server on GNU/Linux

By default, client machine will not be able to connect to a Samba server running on GNU/Linux, for two reasons:

- The GNU/Linux firewall will block incoming connections to the Samba server by default.
- SELinux will block client access to the Samba shares by default.

To work around this, you must configure the GNU/Linux firewall to accept incoming connections through the ports commonly used by Samba. You must also configure SELinux to allow client machines to access the Samba shares.

How to allow connections to the Samba shares to go through the firewall on CentOS 6, RedHat Enterprise Linux 6 or other systems using the iptables service

1. Add the ports used by the Samba service to the list of open ports with the following commands:

```
$ sudo iptables -I INPUT -m state --state NEW -p tcp -m tcp --dport 137 -j ACCEPT
$ sudo iptables -I INPUT -m state --state NEW -p tcp -m tcp --dport 138 -j ACCEPT
$ sudo iptables -I INPUT -m state --state NEW -p tcp -m tcp --dport 139 -j ACCEPT
$ sudo iptables -I INPUT -m state --state NEW -p tcp -m tcp --dport 445 -j ACCEPT
$ sudo service iptables save
$ sudo service iptables restart
```

The GNU/Linux firewall should now allow other machines to connect to the NFS shares.

How to allow connections to the Samba shares to go through the firewall on CentOS 7, RedHat Enterprise Linux 7 or other systems using the iptables service

1. Add the Samba service to the list of allowed services with the following commands:

```
$ sudo firewall-cmd --permanent --zone=public --add-service=samba
$ sudo firewall-cmd --reload
```

The GNU/Linux firewall should now allow other machines to connect to the NFS shares.

How to allow connections to the Samba shares to go through SELinux

1. Allow connections to Samba to go through SELinux by entering the following command:

```
$ sudo setsebool -P samba_export_all_ro=1 samba_export_all_rw=1
```

2. Allow connections to each of your shared folders to go through SELinux by entering the following command for each share:

```
$ sudo chcon -t samba_share_t <path to share>
```

For example, if you are configuring a database server with the database file structure, the Harmony binaries and configuration files and one scene data storage space labeled `usadata000`, you would have to run the following command to allow access to each share:

```
$ sudo chcon -t samba_share_t /usr/local/ToonBoomAnimation/harmony  
[Edition]_17  
$ sudo chcon -t samba_share_t /USA_DB  
$ sudo chcon -t samba_share_t /usadata000
```

3. Now, refresh the security context of each of your shared folders by entering the following command for each share:

```
$ sudo restorecon -R <path to share>
```

For example:

```
$ sudo restorecon -R /usr/local/ToonBoomAnimation/harmony[Edition]_17  
$ sudo restorecon -R /USA_DB  
$ sudo restorecon -R /usadata000
```

At this point, SELinux should allow other machines to connect to your Samba shares without issue.

Configuring a GNU/Linux Database Server for Windows Clients

In order to allow Windows clients to connect to your server using the Configuration Wizard, you must fill in the `server.ini` configuration file, which is located in the `etc` subfolder of the Harmony installation package. This file will provide Configuration Wizard with information on the server as well as of each scene data server it is connected to.

How to configure the `server.ini` file on GNU/Linux

1. In a terminal, type in the following command:

```
$ sudo vi /usr/local/ToonBoomAnimation/harmony[Edition]_
17/etc/server.ini
```

2. In `server.ini`, type in the information for the database server as well as the directory name and hostname of every scene data server in your network, like in the following example:

```
[WizardConfig]
ServerName=<server name>
InstallationDrive=C
UsaShare=<harmony share name>
UsadbDrive=C
UsadbShare=<database share name>
```

Replace the tags between `<>` brackets with the appropriate values. Make sure to remove the `<>` brackets from the configuration file as well:

- **<server name>**: The host name of the Harmony database server.
- **<harmony share name>**: The name of the share that gives access to the Harmony binaries and configuration files. By default, this share is named **usa**.
- **<database share name>**: The name of the share that gives access to the database files. By default, this share is named **USA_DB**.

For example, if your database server is called `harmonyserver`, your Harmony installation is shared under the share name **usa** and your Harmony database is shared under the share name **USA_DB**, this would be the proper configuration for this server:

```
[WizardConfig]
ServerName=harmonyserver
InstallationDrive=C
UsaShare=usa
UsadbDrive=C
UsadbShare=USA_DB
```

3. Under the line starting with **UsadbShare**, enter information about each scene data storage space used by your Harmony database, in the following format:

```
FileSystem<file system number>=C <share name> <server name>
```

Replace the tags between <> brackets with the appropriate values. Make sure to remove the <> brackets from the configuration file as well:

- **<file system number>**: The zero-based index of the scene data storage space. For example, the first scene data storage space for your database should be named **FileSystem0** and the second one, **FileSystem1**.
- **<share name>**: The name of the share containing the scene data. Usually, this share is either named **USADATA** or **usadata**, and its name may be followed by a zero-padded, zero-based number indicating the index of the scene data storage space. For example, the first scene data storage space could be named **USADATA**, **usadata000**, **usadata001**, etc.
- **<server name>**: The host name of the scene data server. If the scene data storage space is hosted on the same machine as the database, then this host name is the same as the host name for the database. Otherwise, you must specify the name of the machine on which the scene data storage space is located.

For example, if this is the first scene data storage space for your database, it is hosted on a computer named **dataserver** and its name is **usadata000**, the appropriate configuration for this would be as follows:

```
FileSystem0=C usadata000 dataserver
```



IMPORTANT

The variable names and share names in **server.ini** are case sensitive.



NOTE

server.ini also specifies the drive letter on which the software, the database files and scene data files are stored. Since we are configuring a UNIX-based server, there is no actual drive letter involved. Hence, we are using the drive letter **C** as a placeholder so as to respect the expected format of the **server.ini** configuration file, but the value of these parameters is not relevant and will not affect your configuration.

Creating a Scene Data Storage Server on GNU/Linux

You can create additional scene data storage spaces for your Harmony database on machines. The database server will connect to these machines to transmit scene data to the clients, allowing you to expand the storage capacity of your database.

Creating a scene data storage space for your Harmony database requires following these steps:

1. Creating a folder named **usadata** followed by the index number of the storage space at the root of the machine.
2. Sharing the folder via NFS and, if Windows clients must be supported, via Samba as well.
3. Mounting this folder on the database server.
4. If Windows clients must be supported, adding this storage server to the server.ini configuration file.

How to create a scene data storage space

1. In a terminal, enter the following command, where **###** is the padded, zero-based index of the storage space you are creating, replace **###** with **001**:

```
$ sudo mkdir -m 777 /usadata###
```

A folder in which scene data will be stored is created at the root of the filesystem.

How to share the scene data storage space

1. Sharing a scene data storage space is done the exact same way as sharing the server's database file system.
 - To share the scene data storage space for macOS and GNU/Linux based clients, see [Sharing the Harmony Database Files Using NFS on GNU/Linux on page 120](#).
 - To share the scene data storage space for Windows clients, see [Sharing the Harmony Database Files Using Samba on GNU/Linux on page 129](#)

Once this is done, you should already be able to mount the new scene data storage space on the database server as well as on clients.

How to mount the new scene data storage space on a GNU/Linux database server

1. In a terminal, create a folder at the root of the filesystem in which the new scene data storage space

will be mounted with the following command, where **<foldername>** is the name of the scene data directory you created on the scene data server:

```
$ sudo mkdir -m 777 /<foldername>
```

2. open the filesystem table in a text editor by typing the following command:

```
$ sudo vi /etc/fstab
```

3. In the filesystem table, add the following line, replacing **<servername>** with the name of the scene data server and **<foldername>** with the name of the scene data directory you created on this scene data server:

```
<servername>:<foldername> /mnt/<foldername> nfs nfs rw,soft,intr,bg 0 0
```

4. Save and quit.
5. Attempt to mount the entries of the filesystem table:

```
$ sudo mount -a
```

6. To make sure that it worked, try creating a file in the new scene data storage, and verify that it's visible from both machines. To do this:

- On one machine, enter the following command:

```
$ touch /<foldername>/testfile
```

- On the other machine, enter the following command to verify that **testfile** is present

```
$ ls /<foldername>/
```

How to add the scene data storage file to the server.ini configuration file

1. In a terminal, type in the following command:

```
$ sudo vi /usr/local/ToonBoomAnimation/harmony[Edition]_17/etc/server.ini
```

If the file was already configured, it most likely contains configuration in the following format:

```
[WizardConfig]
ServerName=<name of Harmony server>
InstallationDrive=C
UsaShare=usa
UsadbDrive=C
UsadbShare=USA_DB
FileSystem0=C usadata000 <name of first scene data server>
```

2. Under the last line starting with **FileSystem0** which contains information about an existing scene

data storage space, enter the information about your new scene data storage space in the following format:

```
FileSystem<file system number>=C <share name> <server name>
```

Replace the tags between <> brackets with the appropriate values. Make sure to remove the <> brackets from the configuration file as well:

- **<file system number>**: The zero-based index of the scene data storage space. For example, the first scene data storage space for your database should be named **FileSystem0** and the second one, **FileSystem1**.
- **<share name>**: The name of the share containing the scene data. Usually, this share is either named **USADATA** or **usadata**, and its name may be followed by a zero-padded, zero-based number indicating the index of the scene data storage space. For example, the first scene data storage space could be named **USADATA**, **usadata000**, **usadata001**, etc.
- **<server name>**: The host name of the scene data server. If the scene data storage space is hosted on the same machine as the database, then this host name is the same as the host name for the database. Otherwise, you must specify the name of the machine on which the scene data storage space is located.

For example, if this is your second scene data storage space for your database server, it is hosted on a computer named **newdataserver** and is named **usadata001**, its configuration would be as follows:

```
FileSystem1=C usadata001 newdataserver
```



IMPORTANT

The variable names and share names in **server.ini** are case sensitive.



NOTE

The first information for a scene data storage space, **C**, is actually the drive letter on which the scene data storage space is located. Since GNU/Linux does not use drive letters, there is no drive letter for this storage space, and we are only specifying a drive letter to respect the format of the **server.ini** configuration file. This value does not have an effect your server configuration.

Setting Up Static IPs for WebCC on GNU/Linux

It is common to configure server machines to have a static IP. This avoids having to rely on DNS resolution to find the server when its IP address changes, and it also allows you to associate the hostname to the server's static IP address on the machine to skip DNS resolution altogether, increasing network performance.

If your machine is currently connected to a DHCP server that dynamically assigns IP addresses, you can configure your machine to permanently use its current IP address. To do this, you can start by making note of the information for your current network connection, then using these parameters for a permanent IP address.

How to obtain your current network configuration on CentOS 6

1. To obtain your machine's hostname, enter the following command:

```
$ hostname
```

Write down the output value.

2. To obtain the network domain, enter the following command:

```
$ hostname -d
```

Write down the output value.

3. To obtain your machine's default gateway and network device, enter the following command:

```
$ route
```

The output will look like this:

Destination	Gateway	Genmask	Flags	Metric	Ref	Use
Iface						
10.120.0.0	*	<subnet mask>	U	1	0	0
<device name>						
default	<default gateway>	0.0.0.0	UG	0	0	0
<device name>						

One of the rows will have **default** in the destination column. In this row, the name of the device in the **Iface** column is the name of your default network device. Write down the name of that device.

4. To obtain your machine's IP and subnet mask, enter the following command, replacing **<device name>** with the name of the device noted in the previous step:

```
$ nmcli -f all dev list iface <device name>
```

Information about the network interface will appear, in either one of the following formats:

GENERAL.DEVICE:	<device name>
GENERAL.TYPE:	802-3-ethernet
GENERAL.DRIVER:	tg3
GENERAL.HWADDR:	<MAC address>

```

GENERAL.STATE:                connected
CAPABILITIES.CARRIER-DETECT: yes
CAPABILITIES.SPEED:           1000 Mb/s
WIRED-PROPERTIES.CARRIER:    on
IP4-SETTINGS.ADDRESS:         <IPv4 address>
IP4-SETTINGS.PREFIX:          <prefix> (<subnet mask>)
IP4-SETTINGS.GATEWAY:         <default gateway>
IP4-DNS1.DNS:                 <DNS server address>

```

Write down the IPv4 address, the subnet mask, the default gateway as well as every DNS server address listed.



NOTE

You can also obtain the IP address (**inet addr**) and network mask (**Mask**) of your device with the command:

```
$ ifconfig <device name>
```

How to obtain your current network configuration on CentOS 7

1. To obtain your machine's hostname, enter the following command:

```
$ hostname
```

Write down the output value.

2. To obtain the network domain, enter the following command:

```
$ hostname -d
```

Write down the output value.

3. To obtain your machine's default gateway and network device, enter the following command:

```
$ route
```

The output will look like this:

Destination	Gateway	Genmask	Flags	Metric	Ref	Use
Iface						
10.120.0.0	*	<subnet mask>	U	1	0	0
<device name>						
default	<default gateway>	0.0.0.0	UG	0	0	0
<device name>						

One of the rows will have **default** in the destination column. In this row, the name of the device in the **Iface** column is the name of your default network device. Write down the name of that device.

4. To obtain your machine's IP and subnet mask, enter the following command, replacing **<device**

name> with the name of the device noted in the previous step:

```
$ nmcli -p -f IP4 connection show <device name>
```

Information about the network interface will appear. The output will be in this format.

```
=====
Activate connection details (0568a4c4-092a-46e9-9f25-2295a1389d6e)
=====
IP4.ADDRESS[1]:                <IPv4 address>/<prefix>
IP4.GATEWAY:                   <default gateway>
IP4.DNS[1]:                   <DNS server address>
IP4.DOMAIN[1]:                 <search domains>
```

Write down the IPv4 address, the default gateway address, the DNS server address(es) and the search domain(s).

To obtain your subnet mask, enter the following command, replacing **<device name>** with the name of the device noted in the previous step:

```
$ nmcli -p -f DHCP4 connection show <device name> | grep 'subnet_mask'
```

The output should look like this:

```
DHCP4.OPTION[3]:                subnet_mask = <subnet mask>
```

Write down the subnet mask.



NOTE

You can also obtain the IP address (**inet**) and network mask (**netmask**) of your device with the command:

```
$ ifconfig <device name>
```

How to set up a static IP for CentOS 6 and 7

1. Before you do this, make sure you have the following information for your machine:

- Name of default network device (ie: **eth0**, **p3p1**)
- Hostname
- Domain name
- Search domain(s)
- IPv4 address

- Network mask
 - Default gateway address
 - DNS server address(es)
2. Open your machine's network configuration file for editing:

```
$ sudo vi /etc/sysconfig/network
```

3. Configure the file as follows, replacing the bolded text with the appropriate values:

```
NETWORKING=yes  
HOSTNAME=<hostname>.<domain name>
```

4. Open your default network interface's configuration file for editing. This file will be named **ifcfg-** followed by the name of the network interface you noted earlier. For example, if your network interface is named **eth0**, the configuration file will be named **ifcfg-eth0**:

```
$ sudo vi /etc/sysconfig/network-scripts/ifcfg-<device name>
```

5. Configure the file with the appropriate values for the following parameters. If a value is already declared for one of these parameters, replace it. If not, add it to the file:

```
DEVICE="<device name>"  
BOOTPROTO=static  
IPADDR=<IPv4 address>  
NETMASK=<network mask>  
GATEWAY=<default gateway>  
ONBOOT=yes
```

6. Open the DNS configuration file for editing:

```
$ sudo vi /etc/resolv.conf
```

7. Configure the file as follows, replacing the bolded text with the appropriate values:

```
search <search domain names>  
nameserver <DNS server address>
```



NOTES

- If you have several search domains, enter them all on the same line after **search**, separated by spaces.
- If you have several DNS servers, add one **nameserver** line for each address.

8. Restart the network service:

```
$ sudo service network restart
```

Checking the Database Server Log on GNU/Linux

In order to troubleshoot problems with the Harmony database server, you can look through its log. This is done differently depending on whether you installed the database server as a daemon or using Service Launcher.

If your server is configured for Windows clients, you can also check the link server log for issues.

How to check the log for the database server daemon

1. In a terminal, enter the following command:


```
$ less /tmp/tbdbserver.log
```

How to check the log for the link server daemon


1. In a terminal, enter the following command:

```
$ less /tmp/Link_srv.log
```

How to check the log for the database server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the system tray, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **Database Server** tab.
4. In the bottom half, select the **Log** tab.
5. Click on the **View Complete Log** button in the bottom-right corner.

How to check the log for the link server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the system tray, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **Link** tab.
4. In the bottom half, select the **Log** tab.
5. Click on the **View Complete Log** button in the bottom-right corner.

Launching the Database Server Manually in GNU/Linux

You can start the database server directly from a terminal. This is usually not practical for a production environment, but it can be very useful when troubleshooting problems.

If your server needs to handle Windows clients, you can also launch the link server in a different terminal to ensure that both services work.

You can view the output of both services directly in the terminal in which you launch them as they start and handle requests.

How to start the database server manually

1. In a terminal, enter the following command:

```
$ /usr/local/ToonBoomAnimation/harmony[Edition]_17/lnx86_64/bin/tbdbserver
```

How to start the link server manually

1. In another terminal, enter the following command:

```
$ /usr/local/ToonBoomAnimation/harmony[Edition]_17/lnx86_64/bin/Link_srv
```

Stopping and Restarting the Database Server on GNU/Linux

If you installed the database server as a daemon, you can stop it and restart it using the **service** command line tool. If you installed it using Service Launcher, you can stop it and restart it manually in Service Launcher.

If you have installed the link server so as to be able to handle Windows clients, stopping and restarting it may also be necessary for the intended purpose.

How to stop and restart the database server daemon

1. In a terminal, enter the following command:

```
$ sudo service toonboom_dbserver stop
```

2. When you are ready to start the server again, enter the following command:

```
$ sudo service toonboom_dbserver start
```



TIP

You can also restart the service instantly with the following command:

```
$ sudo service toonboom_dbserver restart
```

How to stop and restart the link server daemon

1. In a terminal, enter the following command:

```
$ sudo service toonboom_link_srv stop
```

2. When you are ready to start the server again, enter the following command:

```
$ sudo service toonboom_link_srv start
```




TIP

You can also restart the service instantly with the following command:

```
$ sudo service toonboom_link_srv restart
```

How to stop and restart the database server in Service Launcher


1. Make sure you are currently signed in as the user running the service.

2. In the system tray, click on the Service Launcher  icon.

The Service Launcher application opens.

3. Select the **Database Server** tab.
4. Click on **Stop**.
5. When you are ready to start the server again, click on **Launch**.

How to stop and restart the link server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the system tray, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **Link Server** tab.
4. Click on **Stop**.
5. When you are ready to start the server again, click on **Launch**.

Pausing and Resuming the Database Server on GNU/Linux

If you need to perform maintenance or updates on a Harmony server, you can pause it instead of stopping it completely.

Users who attempt to save a scene while the server is paused will receive a warning message. The server will queue the save and commit the changes when the server is resumed.

How to pause the database server

1. In a command prompt, enter the following command:

```
$ /usr/local/ToonBoomAnimation/harmony[Edition]_17/lnx86_64/bin/dbu -p
```

How to check pause status of the database server

1. In a command prompt, enter the following command:

```
$ /usr/local/ToonBoomAnimation/harmony[Edition]_17/lnx86_64/bin/dbu -P
```

How to resume the database server

1. In a command prompt, enter the following command:

```
$ /usr/local/ToonBoomAnimation/harmony[Edition]_17/lnx86_64/bin/dbu -  
res
```

Uninstalling the Database Server on GNU/Linux

If you installed the database server as a daemon, you can uninstall it using the `chkconfig` command line tool. If you installed it using Service Launcher, you can uninstall it in Service Launcher.

How to disable the database server daemon

1. In a terminal, type in the following commands to stop, then disable the daemon:

```
$ sudo service toonboom_dbserver stop
$ sudo chkconfig toonboom_dbserver off
```

The database server daemon will stop and will no longer launch on boot.


How to disable the link server daemon

1. In a terminal, type in the following commands to stop, then disable the daemon:

```
$ sudo service toonboom_link_srv stop
$ sudo chkconfig toonboom_link_srv off
```


The link server daemon will stop and will no longer launch on boot.

How to uninstall the database server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the system tray, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **Database Server** tab.
4. Click on **Stop**.
5. Uncheck the **Launch when Service Launcher starts** option.
6. In the top menu, select **Server > Save All Settings**.

How to uninstall the link server in Service Launcher

1. Make sure you are currently signed in as the user running the service.

2. In the system tray, click on the Service Launcher  icon.

The Service Launcher application opens.

3. Select the **Link Server** tab.
4. Click on **Stop**.
5. Uncheck the **Launch when Service Launcher starts** option.
6. In the top menu, select **Server > Save All Settings**.

Connecting Client Workstations to a Harmony Database Server

On a Harmony database server, database files and scene data files are present and sorted using a specific file structure. To connect a workstation to a Harmony database server, the structure of the database file system must be simulated on the client machine using file sharing.

A client workstation configuration is permanent, meaning that it is only necessary to configure a client to connect to a specific Harmony database server once. Once a client workstation is properly configured, you can simply launch Harmony or any other Harmony application at any time and connect to the server using your Harmony database account.

Configuring Harmony Server Clients on Windows

Provided that your server is properly configured, a Windows machine can be configured to connect to a Harmony database server using Configuration Wizard. The wizard creates Windows shortcut (.lnk) files pointing to the different network shares used for the database on the hard drive of the client machine, and configures a script that signs in to those network shares on boot.


Those Windows shortcuts have the same name as the share they connect to and are placed at the root of your storage drive, and Harmony interprets those shortcuts as if they were directories, a lot like the way symbolic links work on UNIX systems.

Connecting a Windows Client to a Database Server Using Configuration Wizard	152
Manually Configuring a Windows Client to Connect to a Harmony Database	153
Configuring the Hosts File for a Windows Client	156
About Connecting to a Harmony Database Without Using Shortcut Files on Windows	157
Configuring the Shortcuts File on Windows	158
Setting a Custom Location for the Shortcuts Configuration File on Windows	160
Troubleshooting Client Connections on Windows	162

Connecting a Windows Client to a Database Server Using Configuration Wizard

Provided that your server is properly configured, you can quickly configure a Windows workstation to connect to the Harmony server as a client using the Configuration Wizard. You must perform this task for every workstation that will need to connect to your database server.

How to set up a Harmony database client on Windows

1. In the **Start** menu, open the **Programs** menu and select **Harmony 17 [Edition] > Tools >  Configuration Wizard**.
2. When prompted, enable Configuration Wizard to make changes to your device.
The Toon Boom Harmony Configuration Wizard launches.
1. In the Toon Boom Harmony Configuration dialog, select the ... button right of **Database**.
2. In the Toon Boom Harmony Database Creation dialog, select **Client** and click on **Next**.
3. Under **What is the name of the server?**, type in the hostname of the database server.
4. Under **What is the User?**, type in the name of an user account that has access to the database share.
5. Under **What is the Password?**, type in the password for the selected user account.
6. Click on **Next**.
7. In the Database changes field, verify that all the intended changes are correct, then click on **Finish**.

You should now be able to launch Harmony, Control Center, Paint, Scan and Play and connect to the Harmony Server database through these applications.

Manually Configuring a Windows Client to Connect to a Harmony Database

On Windows, the Configuration Wizard uses the `server.ini` file located on the Harmony server to automatically configure clients to connect to each network share used by the database. You can also make this configuration manually. This can be useful if you simply need to add a new scene data storage space to a client, if you made changes to your server configuration or if you need to use different username and password combinations for different network share used by your database, as the Configuration Wizard will use the same credentials for all of them.

Configuring a Windows client is done in two steps:


- Adding the path, username and password to connect to each share used by the Harmony database in the Connections tab of the Harmony Control Panel. Typically, there should be one share named **USA_DB**, which contains the database file structure, and one share named **usadata** followed by a zero-based padded number for each scene data server, such as **usadata000**, **usadata001**, etc.
- Creating a shortcut that points to each of these shares at the root of the main storage drive of your machine.

On Windows, the path to a network share is written as follows:

```
\\servername\sharename
```

This is how the path to each database share must be written in both the Harmony Control Panel and the shortcut files.

How to add Harmony network share connections using the Harmony Control Panel

1. In the **Start** menu, open the **Programs** list, then select **Harmony 17 [Edition] > Tools >  Control Panel**.
2. Confirm that you want to make changes to your device.
The Toon Boom Harmony Control Panel appears.
3. Open the **Connections** tab.
4. In the **Sharename** field, enter the full UNC path of the network share you want to add. For example, if you are creating a connection for **USA_DB** and this network share is located on a machine named **harmony server**, you would type `\\harmonyserver\USA_DB`.
5. In the **Username** field, enter the username of a user who has access to this network share. By default, database files created by the Configuration Wizard can be accessed with the user **usabatch**.
6. In the **Password** field, enter the password of the user account specified in the Username field.
7. Click on **Connect** to verify that the connection can be made using these parameters.

8. Click on **Create Connection** to permanently add this connection to the list of connections that are made on boot.
9. Repeat for each scene data storage space used by the database. For example, if your database has a single scene data storage space named **usadata000** and it is located on the same server as the database files (**harmonyserver**), you would need to add the share **\\harmonyserver\usadata000** using the same credentials.
10. Also repeat for the **usa** share, which is typically located on the same machine as the **USA_DB** share.

How to add shortcuts to the Harmony network shares to your storage drive

1. Go to your desktop by minimizing all application windows.



NOTE

Although the shortcuts must be placed in the root of your storage drive, Windows Explorer will not allow creating shortcuts directly in this location. Therefore, the following steps explain how to create the shortcuts on your desktop, then move them to the root of your storage drive.

2. Right-click anywhere on the desktop, and select **New > Shortcut**.
The Create Shortcut dialog appears.
3. Under **Type the location of the item:**, type the full UNC path of the network share you want to add. For example, if you are creating a shortcut pointing to **USA_DB** and this network share is located on a machine named harmony server, you would type **\\harmonyserver\USA_DB**.
4. Click on **Next**.
5. Under **Type a name for this shortcut:**, type in the name of the share and nothing else. For example, if you are creating a shortcut pointing to **\\harmonyserver\USA_DB**, type in **USA_DB**.
6. Click on **Finish**.
A new shortcut file is created on your desktop.
7. Open an Explorer window.
8. Navigate to the root of the main storage drive for your workstation. Typically, this storage drive is named **C:**
9. Drag and drop the shortcut you just created to the root of your storage drive.
10. When prompted, confirm that you want to provide administrator permission to move the shortcut.

11. Repeat for each scene data storage space used by the database. For example, if your database has a single scene data storage space named **usadata000** and it is located on the same server as the database files (**harmonyserver**), you would need to add another shortcut pointing to **\\harmonyserver\usadata000** with the name **usadata000**.

Configuring the Hosts File for a Windows Client

When a client connects to a server using its hostname, it must first go through the network in order to obtain the IP address associated with that hostname, which adds overhead to the connection.

A hosts file is a configuration file which allows you to forcefully associate certain hostnames with specific IP addresses. This can allow you to set up a client machine to directly connect to the IP address of your Harmony database server without having to query the network for its address first, which may increase network performance. However, this should only be done if your database server has a static IP address which you do not intend to change. If the IP address of your server changes, the information in the host file will no longer be accurate and you will be unable to connect to the server unless you update or remove the configuration in the hosts file.

How to associate the Harmony database server hostname to its IP address

1. In the Start menu, open the Programs list, then select Windows Accessories.
2. Right-click on Notepad and select More > Run as administrator.
3. In the Notepad top menu, select File > Open.
4. In the file type drop-down near the bottom-right corner of the Open dialog, select All Files (*.*)
5. Navigate to `C:\Windows\System32\drivers\etc` and open the file named `hosts`.
6. At the bottom of the file, add a new line.
7. Type in the permanent IP address of the Harmony database server followed by its hostname, separated by a space. For example, if your database server is named `harmonyserver` and its IP address is `10.127.1.1`, you would type in the following:

```
10.127.1.1 harmonyserver
```
8. If your Harmony database server uses other machines for scene data storage, repeat for each machine used for the server so as to associate each machine's hostname to their permanent IP address.
9. In the top menu, select **File > Save**.
10. Reboot the machine.

About Connecting to a Harmony Database Without Using Shortcut Files on Windows

By default, when connecting a client workstation to a Harmony server, shortcut files are placed at the root of the main storage drive (typically **C:**) pointing to the location of each network share used by the Harmony database.

However, it is also possible to configure the location of these network shares in the **shortcuts.conf** configuration file instead of using shortcuts. This lightens the process of accessing database files by removing the overhead from reading the shortcut files.

The **shortcuts.conf** file must be located in the **etc** subfolder of the Harmony installation directory by default, but it can be located anywhere on your computer provided that you specify the location of this file in the **USA_DB_CONFIG** environment variable. This can allow users to change the location of the Harmony database server they are connecting to without needing administrator privileges.

Configuring the Shortcuts File on Windows

The `shortcuts.conf` configuration file should be located in the `etc` subfolder of the Harmony installation directory. It does not exist by default, so it must either be created from scratch, or using the sample file provided with Harmony.

How to create and edit the `shortcuts.conf` configuration file using the sample `shortcuts.conf` file

1. In the Start menu, open the Programs list and select Windows System.
2. Right-click on Command Prompt and select More > Run as administrator
3. Enter the following command to copy the sample `shortcuts.conf` file to its intended location:


```
> CD C:\Program Files (x86)\Toon Boom Animation\Toon Boom Harmony 17 [Edition]
> COPY resources\samples\shortcuts.conf etc\
```
4. Enter the following command to open the `shortcuts.conf` file in Notepad as an administrator:


```
> notepad etc\shortcuts.conf
```

How to fill up the `shortcuts.conf` configuration file

1. If you used the sample `shortcuts.conf` file, the file will already have explanations on lines starting with a `#`, which are ignored by Harmony, followed by the header `shortcut_list :`. If you are creating the `shortcuts.conf` file from scratch, make sure to start by typing in the header:

```
shortcut_list :
```

2. Under `shortcut_list :` add a line specifying the location of the database files (`USA_DB`) in the following format, where `<harmony server>` is the hostname of the computer on which the database files are located:

```
/USA_DB //<server name>/USA_DB
```

For example, if the database files are located on a machine named `harmonyserver`, you would add this line:

```
/USA_DB //harmonyserver/USA_DB
```

3. For each scene data storage used by the Harmony server, add a line in the same format. For

example, if your Harmony database has a scene storage space named `usadata000` on the same machine (`harmonyserver`), and another scene data storage space named `usadata001` on a machine named `scenedataserver`, you would add the following lines:

```
/usadata000 //harmonyserver/usadata000  
/usadata001 //scenedataserver/usadata001
```

4. Save.

The next time you launch Harmony, it will rely on the configuration in `shortcuts.conf` instead of the shortcut files in the root of your storage drive to connect to the database network shares.

Setting a Custom Location for the Shortcuts Configuration File on Windows

By default, `shortcuts.conf` should be located in Harmony's `etc` subfolder, which can only be modified by users with administrator privileges. However, it is possible to specify a for `shortcuts.conf` in the `USADB_CONFIG_FILE` environment variable.

As a result, a user can place a `shortcuts.conf` file in their Documents folder and make Harmony read it, without requiring administrator privileges. Likewise, an administrator can place the `shortcuts.conf` file in a specific location that can be edited by any user, and set the environment variable to point to this file for all users.

How to set a custom location for the `shortcuts.conf` file for the current user

1. First, use a text editor (ie: Notepad) to create a `shortcuts.conf` configuration file in the preferred location—see [Configuring the Shortcuts File on Windows](#).
2. Open the Windows Control Panel:
 - On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
 - On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.
3. Select **User Accounts**, then **User Accounts** again.
4. In the left pane, select **Change my environment variables**.
5. Under the **User variables** list, click on **New**.
The New User Variable dialog appears.
6. In the **Variable Name** field, type in `USADB_CONFIG_FILE`.
7. Click on **Browse File**.
An Open dialog appears.
8. In the Open dialog, navigate to and select the `shortcuts.conf` file you created in the preferred location.
9. In the New User Variable dialog, click on **OK**.
10. In the Environment Variables dialog, click on **OK**.

The next time you launch Harmony, it will use the parameters in the `shortcuts.conf` file in the custom location where you saved it.

How to set a custom location for the `shortcuts.conf` file for all users

1. First, use a text editor (ie: Notepad) to create a `shortcuts.conf` configuration file in the preferred location—see [Configuring the Shortcuts File on Windows](#).
2. Open the Windows Control Panel:
 - On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
 - On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.
3. Select **System and Security > System**.
4. In the left pane, select **Advanced system settings**.
5. Open the **Advanced** tab.
6. Click on **Environment Variables**.
7. Under the **System variables** list, click on **New**.

The New System Variable dialog appears.
8. In the **Variable Name** field, type in `USADB_CONFIG_FILE`.
9. Click on **Browse File**.

An Open dialog appears.
10. In the Open dialog, navigate to and select the `shortcuts.conf` file you created in the preferred location.
11. In the New System Variable dialog, click on **OK**.
12. In the Environment Variables dialog, click on **OK**.

The next time you launch Harmony, it will use the parameters in the `shortcuts.conf` file in the custom location where you saved it.

Troubleshooting Client Connections on Windows

If your client is unable to connect to the database, a good way to diagnose the issue is to verify that you are at least able to connect to and to navigate the database server's shared folders. To test that, you can type the UNC path of one of the database's shared folders in the address bar of an Explorer window. For example, if the database is hosted on a server named **harmonyserver**, try to open the following location:

```
\\harmonyserver\USA_DB
```

Then, you can follow the troubleshooting tips depending on how Windows handled your attempt to connect to the share:

- [Windows Accesses the Database Share Without Issue on page 162](#)
- [Windows Prompts for a Username and/or Password on page 163](#)
- [Windows is Unable to Access the Share and Displays an Error Message on page 163](#)

Windows Accesses the Database Share Without Issue

If you have access to the shared folders, but Harmony is unable to connect to the database, this means the server is properly configured to share files, but not to act as a Harmony database server, either because:

- The database server service is not running on the server machine. You can check if this is the issue by running the server service manually in a terminal on the server machine, then attempting to connect from the client machine again. To do this, depending on the platform of your Harmony server, see:
 - **Windows:** [Launching the Database Server Manually on Windows on page 70](#)
 - **macOS:** [Launching the Database Server Manually in macOS on page 104](#)
 - **GNU/Linux:** [Launching the Database Server Manually in GNU/Linux on page 144](#)



NOTE

Running the database server service directly from a terminal also allows you to immediately see if the service encounters any errors when launching.

- The server machine is refusing connections to the Harmony database server because of its firewall rules. To resolve this, see:
 - **Windows:** [Opening the Firewall Ports for a Harmony Database Server on Windows on page 57](#)
 - **GNU/Linux:** [Opening the Firewall Ports for a Harmony Database Server on GNU/Linux on page 119](#) and [Opening the Firewall Ports for the Link Server on GNU/Linux on page 128](#).



NOTE

The macOS firewall is not known to block connections to a Harmony server by default.

Windows Prompts for a Username and/or Password

If you get prompted for a username or a password, this means that the server is properly configured to share its files, but Harmony failed to establish a connection to the shared folders when you logged in. This may be because:

- Harmony was not configured to use the right credentials to access the shares.

When you use Configuration Wizard to configure a client to connect to a database server, you are prompted to type in a username and password to access the database server's shared folders. Configuration Wizard saves those credentials and, when you log in to your Windows account, Harmony attempts to establish a connection with those shared folders using the username and the password you entered.

You can check if this is the issue by attempting to configure your client again using Configuration Wizard—see [Connecting a Windows Client to a Database Server Using Configuration Wizard on page 152](#).

Windows is Unable to Access the Share and Displays an Error Message

If you are getting an error message, this means that your client is either unable to find the shared folder on your server, or that your server is refusing to let your client access it. This may be because:

- There is something wrong with the configuration of one or several of the database shares. Make sure you configured the shared folders properly on your server:
 - **Windows:** The shares on the actual Harmony database server are configured automatically by Configuration Wizard. For shares on machines other than the Harmony database server, see [Creating a Scene Data Storage Space on Windows on page 59](#).
 - **macOS:** [Sharing the Harmony Database Files Using Samba on macOS on page 89](#) and [Creating a Scene Data Storage Server on macOS on page 95](#).
 - **GNU/Linux:** [Sharing the Harmony Database Files Using Samba on GNU/Linux on page 129](#) and [Creating a Scene Data Storage Server on GNU/Linux on page 135](#).
- If the server is running GNU/Linux, the GNU/Linux firewall and/or SELinux may be blocking incoming connections to its Samba shares—see [Opening the Firewall Ports and Configuring SELinux for a Samba Server on GNU/Linux on page 131](#).
- If your client is using Windows 10, the problem might be caused by the version of the Samba protocol the server is using. Servers that use Samba to share files usually use the SMB 2.0, SMB 2.1 or SMB 3.0 protocol. However, some Samba servers may still be using the deprecated SMB 1.0 protocol. By default, Windows 10 computers are configured to be unable to connect to Samba servers that use the SMB 1.0 protocol, because SMB 1.0 is considered unsafe.

There are three common cases where a Samba server will use the SMB 1.0 protocol:

- The server runs CentOS 6.
- The server runs macOS, and Samba was installed through third party means, such as brew or SMBup.
- The server runs Windows 10, and has the optional SMB 1.0/CIFS Server component enabled.

The optimal solution to this issue is to configure the server to only use SMB 2.0 and up. This is possible in the three cases above.

The alternative solution is to configure your Windows 10 clients to be able to connect to Samba servers that use SMB 1.0. This should only be done if it is not possible to use SMB 2.0 and up across the network, as SMB 1.0 is considered unsafe.

How to configure a GNU/Linux CentOS 6 server to use SMB 2.0 and up

1. In a terminal, type in the following command:

```
$ sudo vi /etc/samba/smb.conf
```

2. Inside the file, scroll down to the section that starts with `[global]`.
3. Press the `i` key to enter insert mode.
4. Add the following line to the `[global]` section:

```
protocol = SMB2
```

5. Press Esc, then type `:wq` and press Enter/Return to save and quit.
6. In the terminal, restart the Samba server with the following command:

```
$ sudo service smb restart
```

From now on, your Windows 10 client should be able to connect to your GNU/Linux Samba server.

How to configure a macOS server to use SMB 2.0 and up

1. In a terminal, type in the following command:

```
$ sudo vi /opt/local/etc/samba3/smb.conf
```

2. Inside the file, scroll down to the section that starts with `[global]`.
3. Press the `i` key to enter insert mode.
4. Add the following line to the `[global]` section:

```
protocol = SMB2
```

5. Press Esc, then type `:wq` and press Enter/Return to save and quit.
6. In the terminal, restart the Samba server with the following command:

```
$ sudo launchctl unload /Library/LaunchDaemons/org.samba.smbd.plist
$ sudo launchctl load /Library/LaunchDaemons/org.samba.smbd.plist
```

From now on, your Windows 10 client should be able to connect to your macOS Samba server.

How to configure a Windows 10 server to use SMB 2.0 and up

1. Open the Windows Control Panel:
 - On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
 - On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.
2. In the Control Panel, select **Programs > Turn Windows features on or off**.
3. In the features list, scroll to and expand **SMB 1.0/CIFS File Sharing Support**.
4. Under SMB 1.0/CIFS File Sharing Support, make sure that the item **SMB 1.0/CIFS Server** is **unchecked**.
5. Click **OK**.

You may be prompted to restart the machine for the changes to take effect. Afterwards, your Windows 10 clients should be able to connect to your Windows shares.

How to configure a Windows 10 client to connect to SMB 1.0/CIFS servers

1. Open the Windows Control Panel:
 - On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
 - On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.
2. In the Control Panel, select **Programs > Turn Windows features on or off**.
3. In the features list, scroll to and expand **SMB 1.0/CIFS File Sharing Support**.
4. Under SMB 1.0/CIFS File Sharing Support, make sure that the item **SMB 1.0/CIFS Client** is **checked**.
5. Click **OK**.

You may be prompted to restart the machine for the changes to take effect. Afterwards, your Windows 10 client should be able to connect to your server's Samba shares.

Configuring Harmony Server Clients on macOS

On macOS and GNU/Linux, a client workstation is configured to connect to a Harmony server by mounting the different network shares used for the database on directories with the same name as these shares, at the root of the filesystem. As a result, all of the database and scene data files become part of your machine's file system, and the path to each file in your database is the same path on both the client machine and the server machine, allowing seamless access to all of your scene data on any machine. The most straightforward approach to this is to configure these shares to be automatically mounted on boot using the file system table (**fstab**) file.

Connecting a macOS Client to a Database Server	167
Setting the User File Creation Permission Mask on macOS	169
Configuring the Hosts File for a macOS Client	170
About the Location of Mount Points on macOS	171
Configuring the Shortcuts File on macOS	172
Setting a Custom Location for the Shortcuts Configuration File on macOS	173

Connecting a macOS Client to a Database Server

The most simple way of connecting a macOS client to a Harmony database server is by mounting the database network shares using the NFS protocol. This can be done automatically on boot by adding entries for each database network share in the file system table (**fstab**).

How to mount Harmony database network shares using the file system table

1. In a terminal, open the file system table for editing:

```
$ sudo vi /etc/fstab
```

2. Add an entry in the following format for each network share that your Harmony database requires:

```
<server name>:<share name> /<share name> nfs rw,soft,intr,rsvport
0 0
```

For example, if your server is named `harmonyserver` and hosts the database files (`USA_DB`) as well as the first scene data storage space, named `usadata000`, and you have another scene data server called `scenedataserver` hosting the second scene data storage space, named `usadata001`, you would need to add the following lines:

```
harmonyserver:/USA_DB      /USA_DB      nfs rw,soft,intr,rsvport 0 0
harmonyserver:/usadata000 /usadata000 nfs rw,soft,intr,rsvport 0 0
scenedataserver:/usadata001 /usadata001 nfs rw,soft,intr,rsvport 0 0
```

3. Create a mount point at the root of your filesystem for each share that you intend to mount:

```
$ sudo mkdir -m 777 /<share name 1> /<share name 2> /<share name 3>
```

For example, the mount points for the `fstab` entries in the previous examples would be created with this command:

```
$ sudo mkdir -m 777 /USA_DB /usadata000 /usadata001
```

4. Attempt to mount the new `fstab` entries with the following command:

```
$ sudo mount -a
```

There will be no output if the command was successful.

5. Verify that you can view the files in each mounted share with the following command:

```
$ ls /<share name>
```

For example:

```
$ ls /USA_DB
$ ls /usadata000
$ ls /usadata001
```


If you can see the database files in your client filesystem, you can now launch Harmony or any other Harmony application and connect to the database.

Setting the User File Creation Permission Mask on macOS

By default, files created on macOS in directories that are accessible to everyone, such as Harmony database directories, are created with a file permission mode that only allows the owner (creator) of the file to modify the file.

Since a Harmony database is a collaborative work environment where everyone needs to be able to make changes to scenes, this must be circumvented by setting the permission mask for file creation (referred to as `umask` or `user mask`). By setting a permissive `umask` on your machine, all the files created by your client on the Harmony database will be modifiable by anyone else with access to the database.

The method to change the user mask has changed as of Mac OS X 10.11. Below are the steps to accomplish this depending on your version of macOS.

How to configure a permissive user mask on Mac OS X 10.11 and later

1. In a terminal, type the following command:

```
$ sudo launchctl config user umask 000
```

2. Reboot the machine.

How to configure a permissive user mask on Mac OS X 10.10 and earlier

1. In a terminal, type the following command to edit your `launchd` configuration file:

```
$ sudo vi /etc/launchd.conf
```

2. Add the following line at the end of the file:

```
umask 0
```

3. Save the file.
4. Reboot the machine.

Configuring the Hosts File for a macOS Client

When a client connects to a server using its hostname, it must first go through the network in order to obtain the IP address associated with that hostname, which adds overhead to the connection.

A hosts file is a configuration file which allows you to forcefully associate certain hostnames with specific IP addresses. This can allow you to set up a client machine to directly connect to the IP address of your Harmony database server without having to query the network for its address first, which may increase network performance. However, this should only be done if your database server has a static IP address which you do not intend to change. If the IP address of your server changes, the information in the host file will no longer be accurate and you will be unable to connect to the server unless you update or remove the configuration in the hosts file.

How to associate the Harmony database server hostname to its IP address

1. In a terminal, open the hosts file for editing:

```
$ sudo vi /etc/hosts
```

2. At the bottom of the file, add a new line.
3. Type in the permanent IP address of the Harmony database server followed by its hostname, separated by a space. For example, if your database server is named **harmonyserver** and its IP address is **10.127.1.1**, you would type in the following:

```
10.127.1.1 harmonyserver
```

4. If your Harmony database server uses other machines for scene data storage, repeat for each machine used for the server so as to associate each machine's hostname to their permanent IP address.
5. Save and quit.

About the Location of Mount Points on macOS

By default, Harmony database shares must be created and mounted at the root of the filesystem. If preferred, it is possible to mount the database shares in different locations, and configure Harmony to look in these locations instead. This is done by creating a **shortcuts.conf** configuration file for Harmony, which will redirect requests to traditional database locations such as **/USA_DB** to a directory of your choosing on your filesystem.

The **shortcuts.conf** file must be located in the **etc** subfolder of the Harmony installation directory by default, but it can be located anywhere on your computer provided that you specify the location of this file in the **USA_DB_CONFIG** environment variable. This can allow users to change the location of the Harmony database server they are connecting to without needing administrator privileges.

Configuring the Shortcuts File on macOS

The `shortcuts.conf` configuration file should be located in the `etc` subfolder of the Harmony installation directory. It does not exist by default.

How to create and edit the `shortcuts.conf` configuration file

1. In a terminal, enter the following command:

```
$ vi "/Applications/Toon Boom Harmony 17 [Edition]/Harmony
[Edition].app/Contents/tba/etc/shortcuts.conf"
```

This will create an empty `shortcuts.conf` file and open it in your command line interface text editor.

2. At the top of the file, type the following header line:

```
shortcut_list :
```

3. Under `shortcut_list :`, add a line specifying the location of the mount point for `USA_DB` on your filesystem in the following format, `<path_to_usa_db>` is the absolute path to the mount point of `USA_DB` on your filesystem, without a trailing slash, and including the `USA_DB` part:

```
/USA_DB /<path_to_usa_db>
```

For example, if the mount point to `USA_DB` is a directory named `USA_DB` in `/Users/jdoe/Documents`, you would add this line:

```
/USA_DB /Users/jdoe/Documents/USA_DB
```

4. For each scene data storage space used by the Harmony server, add a line in the same format. For example, if your Harmony server has two scene storage spaces named `usadata000` and `usadata001`, and their mount points are in the same directory as `USA_DB`, you would add the following lines:

```
/usadata000 /Users/jdoe/Documents/usadata000
/usadata001 /Users/jdoe/Documents/usadata001
```

5. Save and quit.

The next time you launch Harmony, it will look for the database files in the locations specified in `shortcuts.conf`.

Setting a Custom Location for the Shortcuts Configuration File on macOS

By default, `shortcuts.conf` should be located in Harmony's `etc` subfolder, which can only be modified by users with administrator privileges. However, it is possible to specify a for `shortcuts.conf` in the `USADB_CONFIG_FILE` environment variable.

As a result, a user can place a `shortcuts.conf` file in their Documents folder and make Harmony read it, without requiring administrator privileges. Likewise, an administrator can place the `shortcuts.conf` file in a specific location that can be edited by any user, and set the environment variable to point to this file for all users.

How to set a custom location for the `shortcuts.conf` file

1. First, create a `shortcuts.conf` configuration file in the preferred location—see [Configuring the Shortcuts File on macOS](#).
2. Then, do one of the following:

- If you want to set the location of `shortcuts.conf` for your user only, enter the following command in a terminal:

```
$ vi ~/.bash_profile
```

- If you want to set the location of `shortcuts.conf` for all users, enter the following command in a terminal:

```
$ sudo vi /etc/profile
```

You are now editing a script that either runs when your user logs in, or when your machine boots.

3. At the end of the script, add the following lines, replacing `<path to shortcuts.conf>` with the absolute (full) path to the `shortcuts.conf` file you created:

```
export USADB_CONFIG_FILE="<path to shortcuts.conf>"
launchctl setenv USADB_CONFIG_FILE "<path to shortcuts.conf>"
```

4. Save and quit.
5. Reboot.

Configuring Harmony Server Clients on GNU/Linux

On macOS and GNU/Linux, a client workstation is configured to connect to a Harmony server by mounting the different network shares used for the database on directories with the same name as these shares, at the root of the filesystem. As a result, all of the database and scene data files become part of your machine's file system, and the path to each file in your database is the same path on both the client machine and the server machine, allowing seamless access to all of your scene data on any machine. The most straightforward approach to this is to configure these shares to be automatically mounted on boot using the file system table (**fstab**) file.

Connecting a GNU/Linux Client to a Database Server	175
Setting the User File Creation Permission Mask on macOS	177
Configuring the Hosts File for a GNU/Linux Client	179
About the Location of Mount Points on GNU/Linux	180
Configuring the Shortcuts File on GNU/Linux	181
Setting a Custom Location for the Shortcuts Configuration File on GNU/Linux	182

Connecting a GNU/Linux Client to a Database Server

The most simple way of connecting a GNU/Linux client to a Harmony database server is by mounting the database network shares using the NFS protocol. This can be done automatically on boot by adding entries for each database network share in the file system table (**fstab**).

How to mount Harmony database network shares using the file system table

1. In a terminal, enable the NFS and NFS lock daemons:

```
$ sudo chkconfig nfs on
$ sudo service nfs start
$ sudo chkconfig nfslock on
$ sudo service nfs start
```

1. Open the file system table for editing:

```
$ sudo vi /etc/fstab
```

2. Add an entry in the following format for each network share that your Harmony database requires:

```
<server name>:/<share name> /<share name> nfs rw,soft,intr,bg 0 0
```

For example, if your server is named `harmonyserver` and hosts the database files (`USA_DB`) as well as the first scene data storage space, named `usadata000`, and you have another scene data server called `scenedataserver` hosting the second scene data storage space, named `usadata001`, you would need to add the following lines:

```
harmonyserver:/USA_DB /USA_DB nfs rw,soft,intr,bg 0 0
harmonyserver:/usadata000 /usadata000 nfs rw,soft,intr,bg 0 0
scenedataserver:/usadata001 /usadata001 nfs rw,soft,intr,bg 0 0
```

3. Create a mount point at the root of your filesystem for each share that you intend to mount:

```
$ sudo mkdir -m 777 /<share name 1> /<share name 2> /<share name 3>
```

For example, the mount points for the `fstab` entries in the previous examples would be created with this command:

```
$ sudo mkdir -m 777 /USA_DB /usadata000 /usadata001
```

4. Attempt to mount the new `fstab` entries with the following command:

```
$ sudo mount -a
```

There will be no output if the command was successful.

5. Verify that you can view the files in each mounted share with the following command:

```
$ ls /<share name>
```

For example:


```
$ ls /USA_DB  
$ ls /usadata000  
$ ls /usadata001
```

If you can see the database files in your client filesystem, you can now launch Harmony or any other Harmony application and connect to the database.

Setting the User File Creation Permission Mask on macOS

By default, files created on GNU/Linux in directories that are accessible to everyone, such as Harmony database directories, are created with a file permission mode that only allows the owner (creator) of the file and the group they belong to to modify the file.

Since a Harmony database is a collaborative work environment where everyone needs to be able to make changes to scenes, this must be circumvented by setting the permission mask for file creation (referred to as `umask` or `user mask`). By setting a permissive `umask` on your machine, all the files created by your client on the Harmony database will be modifiable by anyone else with access to the database.

On GNU/Linux, you can make sure the user mask is always permissive by making it permissive every time you log in. This is done by adding the command to set the user mask in a log-in script. You can choose whether to use the log-in script for all users or the one for your current user only.

How to configure a permissive user mask for all users on GNU/Linux

1. In a terminal, edit the login script for all users logging in through the Bash shell with the following command:

```
$ sudo vi /etc/bashrc
```

2. In the script file, find the following block:

```
# By default, we want umask to get set. This sets it for non-login shell.
# Current threshold for system reserved uid/gids is 200
# You could check uidgid reservation validity in
# /usr/share/doc/setup-*/uidgid file
if [ $UID -gt 199 ] && [ "`/usr/bin/id -gn`" = "`/usr/bin/id -un`" ]; then
    umask 002
else
    umask 022
fi
```

3. In the block, replace `umask 002` with `umask 000`:

```
# By default, we want umask to get set. This sets it for non-login shell.
# Current threshold for system reserved uid/gids is 200
# You could check uidgid reservation validity in
# /usr/share/doc/setup-*/uidgid file
if [ $UID -gt 199 ] && [ "`/usr/bin/id -gn`" = "`/usr/bin/id -un`" ]; then
    umask 000
else
    umask 022
fi
```

4. Save and quit.
5. Create a `umask` script for all users logging in through a CSH shell with the following command:

```
$ sudo vi /etc/csh.cshrc
```

6. In the script file, find the following block:

```
# By default, we want this to get set.
# Even for non-interactive, non-login shells.
# Current threshold for system reserved uid/gids is 200
# You could check uidgid reservation validity in
# /usr/share/doc/setup-*/uidgid file
if ($uid > 199 && "`/usr/bin/id -gn`" == "`/usr/bin/id -un`) then
    umask 002
else
    umask 022
endif
```

7. In this block, replace **umask 002** with **umask 000**:

```
# By default, we want this to get set.
# Even for non-interactive, non-login shells.
# Current threshold for system reserved uid/gids is 200
# You could check uidgid reservation validity in
# /usr/share/doc/setup-*/uidgid file
if ($uid > 199 && "`/usr/bin/id -gn`" == "`/usr/bin/id -un`) then
    umask 000
else
    umask 022
endif
```

How to configure a permissive user mask for the current user on GNU/Linux

1. In a terminal, open your Bash log-in script for editing with the following command:

```
$ vi ~/.bashrc
```

2. Add the end of the log-in script, add the following command:

```
umask 0
```

3. Save and quit.
4. Open your CSH log-in script for editing with the following command:

```
$ vi ~/.cshrc
```

5. Add the end of the log-in script, add the following command:

```
umask 0
```

Configuring the Hosts File for a GNU/Linux Client

When a client connects to a server using its hostname, it must first go through the network in order to obtain the IP address associated with that hostname, which adds overhead to the connection.

A hosts file is a configuration file which allows you to forcefully associate certain hostnames with specific IP addresses. This can allow you to set up a client machine to directly connect to the IP address of your Harmony database server without having to query the network for its address first, which may increase network performance. However, this should only be done if your database server has a static IP address which you do not intend to change. If the IP address of your server changes, the information in the host file will no longer be accurate and you will be unable to connect to the server unless you update or remove the configuration in the hosts file.

How to associate the Harmony database server hostname to its IP address

1. In a terminal, open the hosts file for editing:

```
sudo vi /etc/hosts
```

2. At the bottom of the file, add a new line.
3. Type in the permanent IP address of the Harmony database server followed by its hostname, separated by a space. For example, if your database server is named **harmonyserver** and its IP address is **10.127.1.1**, you would type in the following:

```
10.127.1.1 harmonyserver
```

4. If your Harmony database server uses other machines for scene data storage, repeat for each machine used for the server so as to associate each machine's hostname to their permanent IP address.
5. Save and quit.

About the Location of Mount Points on GNU/Linux

By default, Harmony database shares must be created and mounted at the root of the filesystem. If preferred, it is possible to mount the database shares in different locations, and configure Harmony to look in these locations instead. This is done by creating a **shortcuts.conf** configuration file for Harmony, which will redirect requests to traditional database locations such as **/USA_DB** to a directory of your choosing on your filesystem.

The **shortcuts.conf** file must be located in the **etc** subfolder of the Harmony installation directory by default, but it can be located anywhere on your computer provided that you specify the location of this file in the **USA_DB_CONFIG** environment variable. This can allow users to change the location of the Harmony database server they are connecting to without needing administrator privileges.

Configuring the Shortcuts File on GNU/Linux

The `shortcuts.conf` configuration file should be located in the `etc` subfolder of the Harmony installation directory. It does not exist by default.

How to create and edit the `shortcuts.conf` configuration file

1. In a terminal, enter the following command:

```
$ vi /usr/local/ToonBoomAnimation/harmony[Edition]_
17/etc/shortcuts.conf
```

This will create an empty `shortcuts.conf` file and open it in your command line interface text editor.

2. At the top of the file, type the following header line:

```
shortcut_list :
```

3. Under `shortcut_list :`, add a line specifying the location of the mount point for `USA_DB` on your filesystem in the following format, `<path_to_usa_db>` is the absolute path to the mount point of `USA_DB` on your filesystem, without a trailing slash, and including the `USA_DB` part:

```
/USA_DB /<path_to_usa_db>
```

For example, if the mount point to `USA_DB` is a directory named `USA_DB` in `/home/jdoe`, you would add this line:

```
/USA_DB /home/jdoe/USA_DB
```

4. For each scene data storage space used by the Harmony server, add a line in the same format. For example, if your Harmony server has two scene storage spaces named `usadata000` and `usadata001`, and their mount points are in the same directory as `USA_DB`, you would add the following lines:

```
/usadata000 /home/jdoe/usadata000
/usadata001 /home/jdoe/usadata001
```

5. Save and quit.

The next time you launch Harmony, it will look for the database files in the locations specified in `shortcuts.conf`.

Setting a Custom Location for the Shortcuts Configuration File on GNU/Linux

By default, `shortcuts.conf` should be located in Harmony's `etc` subfolder, which can only be modified by users with administrator privileges. However, it is possible to specify a for `shortcuts.conf` in the `USADB_CONFIG_FILE` environment variable.

As a result, a user can place a `shortcuts.conf` file in their Documents folder and make Harmony read it, without requiring administrator privileges. Likewise, an administrator can place the `shortcuts.conf` file in a specific location that can be edited by any user, and set the environment variable to point to this file for all users.

On GNU/Linux, it is common to set environment variables for both the Bash and CSH shells, so that they may be used interchangeably. The following steps cover how to do both.

How to set a custom location for the shortcuts.conf file

1. First, create a `shortcuts.conf` configuration file in the preferred location—see [Configuring the Shortcuts File on GNU/Linux](#).
2. Then, do one of the following:
 - If you are setting the location of `shortcuts.conf` for your user only, enter the following command in a terminal to edit the Bash script that runs when you log in:

```
$ vi ~/.bash_profile
```

- If you are setting the location of `shortcuts.conf` for all users, enter the following command in a terminal to edit the Bash script that runs when any user logs in:
- ```
$ sudo vi /etc/profile
```
3. At the end of the script, add the following line, replacing `<path to shortcuts.conf>` with the absolute (full) path to the `shortcuts.conf` file you created:

```
export USADB_CONFIG_FILE="<path to shortcuts.conf>"
```

4. Save and quit.
  5. Do one of the following:
    - If you are setting the location of `shortcuts.conf` for your user only, enter the following command in a terminal to edit the CSH script that runs when you log in:
- ```
$ vi ~/.login
```
- If you are setting the location of `shortcuts.conf` for all users, enter the following command

in a terminal to edit the CSH script that runs when any user logs in:

```
$ sudo vi /etc/csh.login
```

6. At the end of the script, add the following line, replacing **<path to shortcuts.conf>** with the absolute (full) path to the **shortcuts.conf** file you created:

```
setenv USADB_CONFIG_FILE "<path to shortcuts.conf>"
```

7. Reboot.

Harmony Server Configuration

This section will instruct you on how to configure a Harmony database server.

Using Unicode Characters in Harmony	185
Enabling Unicode Characters in Environment, Job and Scene Names	186
Enabling Unicode in Layer/Node Names	187
Configuring Scene Resolution Presets for aHarmony Database	188
Configuring Resolution Presets for Environments or Jobs	190
About Resolution Presets in XML Format	191

Using Unicode Characters in Harmony

By default, Harmony will only allow you to use the following characters in environment, job, scene and layer names:

- Small and capital letters in the Latin alphabet, excluding accented letters
- Numbers
- Dashes (-), underscores (_), plus signs (+) and equal signs (=)

It is possible, however, to enable the use of most of the characters in the Unicode character range. However, this feature has important limitations that must be taken in account:

- macOS and GNU/Linux machines cannot communicate in Unicode characters over the NFS protocol. Even when using Unicode, characters for certain languages use different encoding on GNU/Linux and macOS. While the Samba file sharing protocol can interpret these differences in encoding in the back-end, NFS does not. Hence, for a macOS server to server GNU/Linux clients or for a GNU/Linux server to server a macOS client, you must either use the Samba protocol to share files, or keep Unicode disabled.
- Scenes with Unicode characters in their names cannot be opened from WebCC.
- A scene with Unicode characters in its name that has been exported from a WebCC server cannot be updated on the client side using the Download Database Changes command and cannot be updated on the server side using the Update Database Scene command.
- The Unity engine does not support Unicode characters. Unicode characters should not be used when using Harmony for gaming.
- Unicode characters should not be used in the names of the root folders of the database file systems (ie: **USA_DB**, **usadata000**, etc.)
- Zip archives may encounter issues with Unicode characters. It is recommended to use the 7z archiving format when archiving database files or scene data.

Enabling Unicode Characters in Environment, Job and Scene Names

You can enable Unicode characters in Control Center, allowing you to create environments, jobs and scenes with names containing Unicode characters.



IMPORTANT

Before enabling Unicode characters for your Harmony database, please see [Using Unicode Characters in Harmony](#) to learn about the limitations of Harmony regarding Unicode characters.

How to enable Unicode characters in Control Center

1. In the top menu, select **Admin** and check the **Allow Unicode Names** option.

Enabling Unicode in Layer/Node Names

To enable the use of Unicode characters in the names of layers and nodes, you must enable the Allow Unicode Names preference in Harmony.

How to enable Unicode characters in Harmony

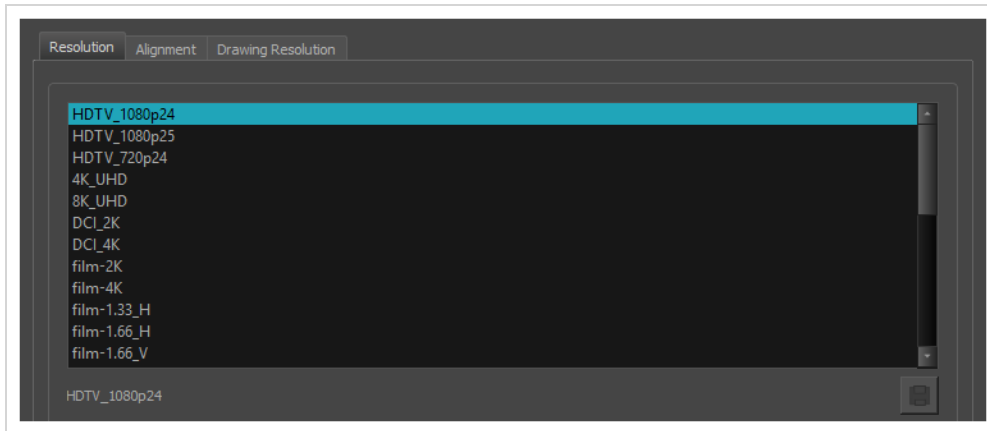
1. In the top menu, select **Edit (Windows/Linux) or Harmony [Edition] (macOS) > Preferences.**

The Preferences dialog opens.

2. Open the **Advanced** tab.
3. Check the **Allow Unicode Names** preference.
4. Click on **OK**.

Configuring Scene Resolution Presets for aHarmony Database

When you create a new database, by default, Harmony's resolution presets are copied to it. These resolution presets can be used by all users of the database:



This file can be found at the following location, and can be edited:

`/USA_DB/resolution/resolution.conf`

If you open the file, you will see some comment lines starting with #, followed by definitions of the resolution presets, like those in the example below:

```
#
# resolution.conf for Harmony Premium
#
# This file defines the scene resolutions available to the program.
#
# The default version of this file can be found in "/usa/etc" for old Harmony/Opus
# or in "<Application Folder>/etc" for Harmony/Animate/Digital Pro/Storyboard.
#
# You may copy this file to /USA_DB/jobs/your_jobs for job specific resolutions
# or /USA_DB/environments/your_environment for environment wide resolutions or /USA_
# DB/resolution/
# for system wide resolutions. In the case of /USA_DB/resolution/ you will have to create
# a folder
# resolution if it doesn't exist already.
#
# You may add any resolution to this file that you frequently use in your scenes.#
#
# syntax:
# resolution <name> <x> <y> [custom] [fps <fps>] [<fov>] [<projection>]
#
# The "custom" keyword means that this resolution was created by a user.
#
# The "fps" keyword can be followed by an integer to specify the number of frames per
# second.
#
# <fov> is either a "V" for vertical fitting, an "H" for horizontal fitting, or an angle
# expressed
# in degrees that specifies a custom field of view. If unspecified, horizontal fitting is
# used.
#
```

```
# <projection> is either "Perspective" or "Orthographic" and is available only in Harmony
# Premium. If unspecified, a perspective projection is used.
#

default HDTV_1080p24
resolution HDTV_1080p24          1920 1080 fps 24 V
resolution HDTV_1080p25          1920 1080 fps 25 V
resolution HDTV_720p24           1280  720 fps 24 V
resolution 4K_UHD                 3840 2160 fps 24 V
resolution 8K_UHD                 7680 4320 fps 24 V
```

As you can see, after the comments, the first line of the file defines which resolution is the default resolution. The default resolution is the resolution new scenes are created with.

```
default <resolution name>
```

Then, each line is used to define one resolution preset, in this format:

```
resolution <resolution name> <width> <height> fps <frame rate> <field of view>
<projection>
```

- The **<resolution name>** is the name of the resolution. This is used to select the resolution preset in the Scene Settings dialog, the default resolution and the resolution in which to render a scene.
- The **<width>** and **<height>** are the dimensions of the scene, in pixels.
- The **<frame rate>** is the amount of frames to play per second.
- The **<field of view>** is the angle of the camera lense. It can be set to **V** to make the camera cone fit the height of the camera field, to **H** to make the camera cone fit the width of the camera field, or to an angle in degrees. If unspecified, the field of view is made to fit horizontally.
- The **<projection>** is how elements are projected in the 3D space. It can either be Perspective or Orthographic. If unspecified, the projection will be set to Perspective.



NOTE

Orthographic projection is only available in Harmony Premium.

Additionally, you can add the **custom** keyword just before **fps** to specify that this resolution is user-created.

Configuring Resolution Presets for Environments or Jobs

It is possible to create resolution preset lists specifically for environment and for jobs. This allows you to make certain resolution presets only available if the scene is for a project that is appropriate for it. To do this, simply create a resolution presets file, either in regular format or XML format, in the appropriate location:

- To create a resolution presets list for an environment, create the list in:

```
/USA_DB/environments/<environment name>/resolution.conf,
```

Replace **<environment name>** with the name of the environment.

- To create a resolution presets list for a job, create the list in:

```
/USA_DB/db_jobs/<job name>/resolution.conf
```

Replace **<job name>** with the name of the job.



NOTE

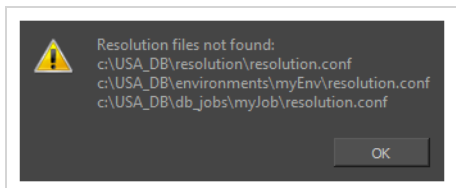
You can also name the file **resolution.xml** and create your resolution presets in XML format—see [About Resolution Presets in XML Format](#).

If several resolution preset lists are available for a scene, for example, if the scene's job, the scene's environment and the database all have their own resolution preset lists, all those lists will be combined, making all of the presets in all of these lists available for the scene.

If the same preset is present in two resolution preset lists, it will only appear once in the list of available resolution presets, so there will not be any duplicate.

If a preset with the same name, but different properties, is listed in more than one of the resolution presets lists, the one at the lowest level will be used. For example, if the resolution preset lists for the scene's job, its environment and its database all contain a preset named **4K**, the preset for the scene's job will be used. If the preset is only in the scene's environment presets list and in database presets list, the one in the environment's presets list will be used.

If you do not want to have scene presets that can be used across all environments and jobs, you can delete the resolution presets file for your database. When you open a scene, Harmony will automatically look for a resolution presets file for the database, for the scene's environment and for the scene's job. If it finds a resolution presets in any of these locations, it will use it. If it does not find resolution presets in any of these three locations, it will display the following error message:



About Resolution Presets in XML Format

By default, resolution presets used by Harmony are stored in a file named **resolution.conf** which defines each preset using a very basic format, where each preset is defined using a line, and each of its properties are separated by spaces.

If you edit or create a resolution preset in Harmony, your resolution presets file will be saved as **resolution.xml**, in the same directory, and the presets will be defined in XML format. From that point on, the **resolution.conf** file will be ignored. Therefore, if you want to manually edit your resolution presets for a database, an environment or a job, and the corresponding directory contains both a **resolution.conf** and a **resolution.xml** file, you must edit the **resolution.xml** file.

The **resolution.xml** file is formatted like in the example below:

```
<!DOCTYPE resolutions>
<!--This file defines the scene resolutions available to the program.-->
<!--The default version of this file can be found in "<Application Folder>/etc" -->
<!--You may copy this file to /USA_DB/jobs/your_job for job specific resolutions-->
<!--or /USA_DB/environments/your_environment for environment wide resolutions.-->
<!--You may add any resolution to this file that you frequently use in your scenes.-->
<!---->
<!--The syntax follows standard xml format, which each tag followed by its value.-->
<!--The "custom" keyword means that this resolution was created by a user.-->
<!--The "FPS" keyword specifies the number of frames per second.-->
<!--The "fovFit" specifies the field of view. It is either vertical, horizontal, or
custom, with the latter expressed-->
<!--as an angle in degrees that specifies a custom field of view. If unspecified,
horizontal fitting is used.-->
<!--The "projectionType" is either "perspective" or "orthographic". If unspecified, a
perspective projection is used.-->
<resolutions>
  <default name="HDTV_1080p24"/>
  <resolution FPS="24" fovFit="vertical" projectionType="perspective" resX="1920"
resY="1080" name="HDTV_1080p24"/>
  <resolution FPS="25" fovFit="vertical" projectionType="perspective" resX="1920"
resY="1080" name="HDTV_1080p25"/>
  <resolution FPS="24" fovFit="vertical" projectionType="perspective" resX="1280"
resY="720" name="HDTV_720p24"/>
  <resolution FPS="24" fovFit="vertical" projectionType="perspective" resX="3840"
resY="2160" name="4K_UHD"/>
  <resolution FPS="24" fovFit="vertical" projectionType="perspective" resX="7680"
resY="4320" name="8K_UHD"/>
  <resolution FPS="24" fovFit="vertical" projectionType="perspective" resX="2048"
resY="1080" name="DCI_2K"/>
  <resolution FPS="24" fovFit="vertical" projectionType="perspective" resX="4096"
resY="2160" name="DCI_4K"/>
</resolutions>
```

As you can see, its format is very similar to resolution.conf, except that it uses an XML structure constructed like so:

```
<!DOCTYPE resolutions>
<resolutions>
```



```
<!--Resolution presets go here-->
</resolutions>
```

Inside the `<resolutions>` block, the first line is a `<default>` tag, which defines the default resolution preset to use when creating a scene, in this format:

```
<default name="preset_name"/>
```

Then, each `<resolution>` tag defines a single resolution preset, in this format:

```
<resolution FPS="frame rate" fovFit="field of view" projectionType="projection type"
resX="width" resY="height" name="preset_name"/>
```

Similar to the lines in `resolution.conf`, each `<resolution>` tag contains the following attributes:

- **FPS:** The amount of frames to play per second.
- **fovFit:** The angle of the camera lense. It can be set to **vertical** to make the camera cone fit the height of the camera field, to **horizontal** to make the camera cone fit the width of the camera field, or to an angle in degrees. If unspecified, the field of view is made to fit horizontally.
- **projectionType:** How elements are projected in the 3D space. It can either be perspective or orthographic. If unspecified, the projection will be set to perspective.



NOTE

Orthographic projection is only available in Harmony Premium.

- **resX:** The width of the scene in pixels.
- **resY:** The height of the scene in pixels.
- **name:** The name of the resolution. This is used to select the resolution preset in the Scene Settings dialog, the default resolution and the resolution in which to render a scene.

Chapter 3: WebCC

WebCC is a web server suite that essentially allows you to host your Harmony database on the Internet, letting freelancers and off-site contributors work on your production from any location with Internet access. Users can browse your Harmony database using a web browser, download scenes, work on the downloaded scenes locally using their copy of Harmony, then upload their changes back to the database.

Because it is entirely web-based, WebCC users do not need to be directly connected to the Harmony server nor to have a Harmony Server license to use WebCC. All they need is a web browser, as well as a licensed copy of Harmony Stand Alone to work on the scenes they download.

WebCC can however also be used on machines that are connected to the Harmony database server, as a complement to Control Center. In this case, scenes can be opened directly from WebCC in Harmony, without having to download them first.

WebCC Server Installation

One of the first things you must know before configuring a WebCC server is that a WebCC server in itself is not a database server, but rather, an addition to a database server. Hence, if you have not already done so, you should follow the steps to create a Harmony database server—see [About Database Server Installation on page 49](#).

Once your Harmony database server is configured, configuring a WebCC server is relatively straightforward. The file structure for the WebCC server is already included in the Harmony installation directory and is ready to use. As soon as you launch the WebCC service on a machine that has access to the database and allow connections to it to go through the firewall, it is already possible to connect to WebCC using a web browser on any machine on the site.



NOTES

- To optimize both security and performance, it may be useful to set up WebCC for internal network access only on the same machine that acts as the Harmony database server, and then install WebCC for World Wide Web access on a dedicated Web server. With this approach, you do not have to give access to the database server to the Web.
- The following instructions will explain how to set up a WebCC server which will be accessible on-site, but will not explain how to make the server accessible off-site.

Installing WebCC on Windows

On Windows, installing a WebCC server with the default parameters can easily be done using the Configuration Wizard.

Disabling the Firewall for a WebCC Server on Windows	197
Installing the WebCC Service on Windows	198
Running the Database Server Service on Log-In on Windows	199
Configuring WebCC with Custom Parameters on Windows	201
About SSL, HTTPS and WebCC on Windows	206
Configuring WebCC to Use SSL on Windows	207
Setting Up Static IPs for WebCC on Windows	211
Opening the Firewall Ports for a Harmony WebCC Server on Windows	215
Checking the WebCC Server Log on Windows	217
Launching the WebCC Server Manually on Windows	218
Stopping and Restarting the WebCC Server on Windows	219
Uninstalling the WebCC Server on Windows	221

Disabling the Firewall for a WebCC Server on Windows

By default, the Windows firewall will block incoming connections to a WebCC server. This installation guide provides instructions on how to add exceptions for the WebCC server to your firewall. However, you may prefer to disable the firewall completely so as to simplify the installation process or to rule out the firewall as a possible cause when troubleshooting an issue.



IMPORTANT

If you choose to disable the firewall on Windows, it is critical to ensure that your domain has effective site-wide security measures to prevent unauthorized access to your WebCC server.

How to disable the firewall on Windows

1. Open the Windows Control Panel:
 - On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
 - On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.
2. In the Control Panel, select **System and Security**.
3. Under **Windows Defender Firewall**, select **Check firewall status**.
4. In the list to the left, select **Turn Windows Defender Firewall on or off**.
5. Under **Domain network settings**, select **Turn off Windows Defender Firewall (not recommended)**.
6. Under **Private network settings**, select **Turn off Windows Defender Firewall (not recommended)**.
7. Under **Public network settings**, select **Turn off Windows Defender Firewall (not recommended)**.
8. Click on **OK**.

Installing the WebCC Service on Windows


You can easily install WebCC as a Windows service using the Configuration Wizard. It will launch as soon as the machine boots, under user **usabatch**.



NOTE

The following steps indicate how to make the WebCC server service launch on boot, via the user **usabatch**, who will have administrator privileges. If you prefer, you can make the database server launch on log-in as the logged-in user. For more information, see [Running the Database Server on Log-In on macOS on page 81](#).

How to install the WebCC service

1. In the **Start** menu, open the **Programs** menu and select **Harmony 17 [Edition] > Tools >  Configuration Wizard**.
2. When prompted, enable Configuration Wizard to make changes to your device.
The Toon Boom Harmony Configuration Wizard launches.
3. In the list of tasks, right of **Harmony Web Control Center Server**, click on
4. In the ensuing dialog, under **Do you wish to install the Harmony Web Control Center Server?**, select **Yes**.
5. Click on **Next**.
6. Click on **Finish** to confirm the changes.

The WebCC service is now installed and should be running. You can test that it works by launching a web browser and going to the following address:

`http://localhost:8080/`

Running the Database Server Service on Log-In on Windows

The traditional way of running a WebCC server is by running the server as a Windows service. This means it launches and allows clients to connect as soon as the machine is booted, without any user being signed-in.

Using Service Launcher, it is possible to configure your WebCC server to only launch as the currently logged-in user, when they log in. This means that WebCC will only be accessible if the user configured to run it is logged in.

Although this may seem impractical, it may be preferable for security reasons. Services installed with the Configuration Wizard are run under the user **usabatch**, an user with administrator privileges which is created by Configuration Wizard to launch the services. When using Service Launcher, the logged-in user does not need to have administrator privileges to run services.


Before configuring Service Launcher to run the WebCC server, the windows service that launches it must be disabled, so as to avoid launching two instances of the WebCC server simultaneously.



NOTE

While Windows services run regardless of who is logged in and even when no user is logged-in, Service Launcher will only run when the user for which it was specifically configured is logged in. This means that if you want Service Launcher to launch when any user is logged in, you must configure it for each user that has access to the machine.

How to disable the WebCC server service

1. Open the **Start** menu and, in the **Programs** menu, select **Harmony 17 [Edition] > Tools >  Control Panel**.
2. When prompted, allow Control Panel to make changes to your device.
3. In the **Process Starter** drop-down, select **HarmonyStarter0-webcc.bat**.



NOTE

The number after **HarmonyStarter** may be different, depending on how many Harmony services were installed before this one.

4. Click on **Stop**.
5. Click on **Remove**.

From that point on, the WebCC server service will no longer launch on boot.

6. Close the **Control Panel**.



How to configure the WebCC server to launch on user log-in

1. Make sure you are currently signed-in as the user who will run the service.



NOTE

This user will need to be logged in for the service to be running. The user does not need to have administrator privileges.

2. Do one of the following:
 - If Service Launcher is not running, open the **Start** menu and, in the **Programs** list, select **Harmony 17 [Edition] > Tools >  Service Launcher**.
 - If Service Launcher is already running, click on its icon  in the notification tray.

The Service Launcher application launches.

3. In the **Application parameters** section of the **Preferences** tab, make sure that the **Launch on login** option is checked, so that services configured in Service Launcher automatically launch when you log in to your account.
4. Select the **WebCC** tab.
5. Check the **Launch when Service Launcher starts** option.
6. To make sure users must enter their password when logging in to WebCC, check the **Authentication** option.
7. By default, WebCC will use port **8080**. If you want to use a specific port to connect to WebCC, type it in the **Port** field.
8. Click on **Launch** to verify that WebCC launches successfully through Service Launcher using the selected parameters.
9. In the top menu, select **Server > Save All Settings**.

From now on, the WebCC server will launch through Service Launcher when you log in.



NOTE

You can enter parameters in the Other Parameters field to customize the WebCC server. For more information, see [Configuring WebCC with Custom Parameters on Windows on page 201](#)

Configuring WebCC with Custom Parameters on Windows

WebCC can run with no parameters, but it also provides some customization options. Among other things, you can change the port number, adjust its security settings and configure it to run with SSL security.


If you installed WebCC as a Windows service, you can customize its parameters by modifying its command line in the Harmony Control Panel. If you use Service Launcher to run WebCC, you can easily add parameters in the Other Parameters field of the WebCC tab of Service Launcher.



TIP

You can also configure WebCC to run a custom script or executable every time a user updates a scene, whether by importing it or by updating it from Harmony. To do this, on the WebCC server, set the environment variable **WEBCC_POST_IMPORT_SCRIPT** to the path of a script or executable that you want to be launched each time a scene is updated. The script or executable will be launched with the environment, job and scene name as its command line parameters, in that order.

How to stop the WebCC server service using the Harmony Control Panel

1. Open the **Start** menu and, in the **Programs** menu, select **Harmony 17 [Edition] > Tools >  Control Panel**.
2. When prompted, allow Control Panel to make changes to your device.
3. In the **Process Starter** drop-down, select **HarmonyStarter0-webcc.bat**.



NOTE

The number after **HarmonyStarter** may be different, depending on how many Harmony services were installed before this one.

4. Click on **Stop**.
5. Click on the Command Line field and press the End key to move the text cursor after **webcc.bat**.
6. After **webcc.bat**, add a space, then type each parameter you want to add, separated by spaces. For example, if you want WebCC to use the port number 8081, you would edit the line like so:

```
C:\Program Files (x86)\Toon Boom Animation\Toon Boom Harmony 17
[Edition]\win64\bin\webcc.bat -port 8081
```




NOTE

For a list of parameters and their usage, see [WebCC Parameters Table on page 202](#).

7. Click on **Update parameters** to save the command line parameters.
8. Click on **Start** to restart WebCC.

How to add parameters for WebCC in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the notification tray, click on the  Service Launcher icon.
The Service Launcher application opens.
3. Select the **WebCC** tab.
4. If you want to let any user connect to WebCC without credentials and have administrative privileges, uncheck the **Authentication** option. Otherwise, make sure it is checked.
5. If you want to change the port used by the WebCC server, type the desired port number in the **Port** field.
6. If you want to specify any other option, type them in the **Other Parameters** field, separated by spaces.



NOTE


For a list of parameters and their usage, see [WebCC Parameters Table on page 202](#).

7. In the top menu, select **Server > Save All Settings**.
8. Click on the **Stop** button.
9. Click on the **Launch** button.

WebCC Parameters Table

The following is a table of parameters that can be specified to customize the behaviour of your WebCC server. They can be used in both the WebCC shell script and in Service Launcher.

Parameter	Description
<code>-numWorkers <number></code>	<p>The amount of server nodes to launch to process client requests. Each node requires one CPU core, and a single node can only process one client request at a time. Therefore, the more nodes are launched, the less likely the server is to stall under heavy load.</p> <p>If this parameter is not specified, WebCC will use as many nodes as there are CPU cores on the server.</p>

Parameter	Description
<code>-noAuth</code>	Disables authentication. With this option enabled, any user accessing the WebCC web interface will be logged in as usabatch and will administrative privileges on the entire database.
<code>-port <port number></code>	<p>The port to which WebCC is deployed.</p> <p>The default port for accessing WebCC is 8080. For browsers to access WebCC without specifying the port number, you can use port 80.</p> <div data-bbox="565 596 1429 764" style="border: 1px solid green; padding: 5px;"> <p> NOTE</p> <p>If either the <code>-http</code> or <code>-https</code> parameter is specified, the <code>-port</code> parameter will be ignored.</p> </div>
<code>-exchange <directory></code>	<p>The location of the exchange directory on the local filesystem. This directory is where all packages are stored when exporting or importing scenes.</p> <p>By default, this directory is <code>/USA_DB/exchange</code>.</p>
<code>-cacheDuration <seconds></code>	The amount of seconds to keep data in the cache. This can be set to any value between 1 and 999999. By default, this is set to 3600.
<code>-allowAllOrigin</code>	<p>Enables cross-origin resource sharing on the WebCC server.</p> <p>By default, as a standard security measure, only files on the same server as the WebCC server are allowed to use WebCC resources. With this option enabled, it is possible to create web interfaces on other servers that use resources stored on the WebCC server, or to store different modules of WebCC on different servers and let them communicate together.</p> <p>In most use cases, this option should be disabled to prevent other machines from exploiting resources on the WebCC server.</p>
<code>-disableAnalytics</code>	Disables Google Analytics tracking for all clients. By default, this is enabled.
<code>-permission <user type> <action> <on off></code>	<p>Overrides a default permission for the given user type.</p> <p>At the moment, this parameter can only be used for the user type animator and for the permission import_export_any_scene. Hence, when used with the following syntax, animator-type users can export and import any scene in the database, without requiring the scene to be assigned to them first.</p> <p style="text-align: center;"><code>-permission animator import_export_any_scene on</code></p>

HTTPS/SSL Parameters Table


The following parameters can be used to enable and configure SSL for your WebCC server.



NOTE

For more information on how to configure WebCC with SSL, see [About SSL, HTTPS and WebCC on Windows on page 206](#).

Parameter	Description
<code>-https <port number></code>	<p>Enables connection to the server through the https protocol using the specified port number.</p> <p>A commonly used port for accessing WebCC through the https protocol is 8443. For web browsers to access WebCC via https without specifying the port number—provided that the address of the server is preceded by https://—you can use port 443.</p> <p>For https to work, you must either specify a pair of PEM format key and certificate with the -key and -cert option, or a PFX format certificate with the -pfx option.</p>
<code>-http <port number></code>	<p>If access through the https protocol is enabled, this parameter allows you to also allow access through the http protocol through the specified port.</p> <p>Using this parameter is optional. If the -https parameter is specified, but not the -http parameter, WebCC will only accept connections through https.</p> <p>The default port for accessing WebCC through the http protocol is 8080. For browsers to access WebCC via http without specifying the port number, you can use port 80.</p>
<code>-key <key file></code>	<p>The location of the private key file for the SSL session. This must be specified along with the -cert parameter.</p> <p>The private key must be in Privacy Enhanced Mail (PEM) format, which is an ASCII format where the key is encoded in Base64 and located between -----BEGIN PRIVATE KEY----- and -----END PRIVATE KEY----- tags. Typically private keys in this format have a .key extension.</p>
<code>-cert <certificate file></code>	<p>The location of the security certificate file for the SSL session. This must be specified along with the -key parameter.</p> <p>The certificate must be in Privacy Enhanced Mail (PEM) format, which is an ASCII format where the certificate is encoded in Base64 and located between -----BEGIN CERTIFICATE----- and -----END CERTIFICATE----- tags. Typically, such certificates have a .crt, .cer or .pem extension.</p>

Parameter	Description
<pre>-pfx <pfx certificate file></pre>	<p>If you have a Personal Information Exchange (PFX) certificate, you can use this parameter to specify its location, instead of using a PEM certificate and key. Typically, such certificates have a <code>.pfx</code> or <code>.p12</code> extension. Since these certificates include their own private key, specifying a key with this certificate type is not required.</p>
<pre>-ca <root ca certificate file></pre>	<p>If you have a domain certificate that is signed by a certificate authority (CA), this parameter allows you to specify the root CA certificate and/or its intermediate certificates.</p> <p>If you want to specify several certificates so as to compose the chain going from your domain certificate to the root CA certificate, you can repeat the parameter, like so:</p> <pre>-ca <intermediate certificate 1> -ca <intermediate certificate 2> -ca <root CA certificate></pre> <p>Typically, you will be able to log on to WebCC using SSL from a Web browser even if WebCC is only configured with your domain certificate, provided that it is signed by a reputable certificate authority. However, if you want project collaborators to be able to use the Download Database Changes and Update Database Scene functionalities in Harmony via SSL, it may be necessary to specify the root CA certificates and/or its intermediates, as Harmony may not otherwise be able to recognize the authority of the CA who signed your certificate.</p> <p>Typically, you can obtain the root CA certificates or intermediate certificates from the root certificate authority of the certificate authority who signed your domain certificate.</p> <div data-bbox="565 1247 1430 1455" style="border: 1px solid green; padding: 10px;"> <p> NOTE</p> <p>If necessary, you can allow Harmony to trust the WebCC server even if it has a self-signed certificate—see Disabling SSL Certificate Validation for WebCC in Harmony on page 280.</p> </div>

About SSL, HTTPS and WebCC on Windows

You can configure your WebCC server to accept connections through the https protocol, either exclusively or alongside the http protocol. Connections through https are encrypted and the identity of the server is verified using a certificate. This ensures the information exchanged between the client and the server cannot be deciphered if intercepted. Also, provided that you configure WebCC with a certificate signed by a reputable certificate authority, this ensures that hackers cannot perform man-in-the-middle attacks by making their own server pose as your WebCC server to intercept its traffic.

For WebCC to accept connections with the https protocol, you must configure WebCC with the following:

- The port used for https connections.
- The private certificate key.
- The certificate.
- Optionally, the root CA and/or intermediate certificate.

There are two types of certificates supported by WebCC:

- **Privacy Enhanced Mail (PEM) certificates:** These certificates are ASCII files encoded in Base64. Typically, the certificate is in a file with a `.crt`, `.cer` or `.pem` extension and the private key is in a file with a `.key` extension.
- **Personal Information Exchange (PFX /PKCS #12) certificates:** These certificates are binary files that include both the certificate and its private key. Typically, such certificates are used for Windows servers and have a `.pfx` or `.p12` extension.

If your certificate is in any other format, such as DER, PKCS #7 or CA_bundle, you will need to perform extra steps to convert the certificate to a supported format.

About Certificate Signature

It is recommended to use a certificate that has been signed by a reputable certificate authority.

If your certificate is signed by a certificate authority, WebCC will typically be accessible without issue in a web browser. However, if you want clients to be able to upload and download scene changes directly between Harmony and the WebCC server, you may have to specify the root and/or intermediate certificate for your certificate. This is because web browsers are typically designed to trace a certificate's chain of trust all the way to the root certificate authority, whereas Harmony is not.

If your certificate is self-signed, web browsers will warn the user that the website is not secure, and users will need to go around that warning to access WebCC. Likewise, Harmony will not be able to upload or download scene changes directly from its interface at all, unless the client makes Harmony trust self-signed certificates by enabling the hidden preference `WEBCC_SSL_SELF_SIGNED`—see [Disabling SSL Certificate Validation for WebCC in Harmony on page 280](#).

Configuring WebCC to Use SSL on Windows

Like other WebCC options, configuring WebCC to use SSL is done by using command line parameters when launching WebCC. To use WebCC with SSL, you must specify the **-https** parameter followed by the port WebCC should use for https connections, the certificate and key file.

The following are examples of parameters to use for configuring WebCC to use SSL:

- If you have a PEM certificate and key file:

```
-https 8443 -key <path to PEM key file> -cert <path to PEM certificate>
```

- If you have a PEM certificate and its private key in the same file, specify the same file using both the **-key** and **-cert** parameters:

```
-https 8443 -key <path to PEM certificate> -cert <path to PEM certificate>
```

- If you have a PFX certificate, simply specify its path with the **-pfx** parameter:

```
-https 8443 -pfx <path to PFX certificate>
```


- If you want to allow users to connect to your server via either http or https, you can do so by specifying a port number for each protocol:

```
-http 8080 -https 8443 -key <path to key file> -cert <path to certificate>
```

- If you have a root certificate, an intermediate certificate, or both, you can specify each file preceded by the **-ca** parameter, in any order:

```
-http 8443 -key <path to key file> -cert <path to certificate> -ca <path to intermediate certificate 1> -ca <path to intermediate certificate 2> -ca <path to root certificate>
```

How to stop the WebCC server service using the Harmony Control Panel

1. Open the **Start** menu and, in the **Programs** menu, select **Harmony 17 [Edition] > Tools >  Control Panel**.
2. When prompted, allow Control Panel to make changes to your device.

3. In the **Process Starter** drop-down, select **HarmonyStarter0-webcc.bat**.

**NOTE**

The number after **HarmonyStarter** may be different, depending on how many Harmony services were installed before this one.

4. Click on **Stop**.
5. Click on the Command Line field and press the End key to move the text cursor after **webcc.bat**.
6. After **webcc.bat**, add a space, then type each parameter you want to add, separated by spaces. For example, if you want WebCC to use the port number 8081, you would edit the line like so:


```
C:\Program Files (x86)\Toon Boom Animation\Toon Boom Harmony 17
[Edition]\win64\bin\webcc.bat -port 8081
```

**NOTE**

For a list of parameters and their usage, see [HTTPS/SSL Parameters Table on page 209](#).

7. Click on **Update parameters** to save the command line parameters.
8. Click on **Start** to restart WebCC.

How to add parameters for WebCC in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the notification tray, click on the  Service Launcher icon.
The Service Launcher application opens.
3. Select the **WebCC** tab.
4. If you want to let any user connect to WebCC without credentials and have administrative privileges, uncheck the **Authentication** option. Otherwise, make sure it is checked.
5. If you want to change the port used by the WebCC server, type the desired port number in the **Port** field.
6. If you want to specify any other option, type them in the **Other Parameters** field, separated by spaces.

**NOTE**


For a list of parameters and their usage, see [HTTPS/SSL Parameters Table](#) on page 209.

7. In the top menu, select **Server > Save All Settings**.
8. Click on the **Stop** button.
9. Click on the **Launch** button.

HTTPS/SSL Parameters Table

The following parameters can be used to enable and configure SSL for your WebCC server.

Parameter	Description
<code>-https <port number></code>	<p>Enables connection to the server through the https protocol using the specified port number.</p> <p>A commonly used port for accessing WebCC through the https protocol is 8443. For web browsers to access WebCC via https without specifying the port number—provided that the address of the server is preceded by https://—you can use port 443.</p> <p>For https to work, you must either specify a pair of PEM format key and certificate with the <code>-key</code> and <code>-cert</code> option, or a PFX format certificate with the <code>-pfx</code> option.</p>
<code>-http <port number></code>	<p>If access through the https protocol is enabled, this parameter allows you to also allow access through the http protocol through the specified port.</p> <p>Using this parameter is optional. If the <code>-https</code> parameter is specified, but not the <code>-http</code> parameter, WebCC will only accept connections through https.</p> <p>The default port for accessing WebCC through the http protocol is 8080. For browsers to access WebCC via http without specifying the port number, you can use port 80.</p>
<code>-key <key file></code>	<p>The location of the private key file for the SSL session. This must be specified along with the <code>-cert</code> parameter.</p> <p>The private key must be in Privacy Enhanced Mail (PEM) format, which is an ASCII format where the key is encoded in Base64 and located between <code>-----BEGIN PRIVATE KEY-----</code> and <code>-----END PRIVATE KEY-----</code> tags. Typically private keys in this format have a <code>.key</code> extension.</p>
<code>-cert <certificate></code>	<p>The location of the security certificate file for the SSL session. This must be</p>

Parameter	Description
<code>file></code>	<p>specified along with the <code>-key</code> parameter.</p> <p>The certificate must be in Privacy Enhanced Mail (PEM) format, which is an ASCII format where the certificate is encoded in Base64 and located between <code>-----BEGIN CERTIFICATE-----</code> and <code>-----END CERTIFICATE-----</code> tags. Typically, such certificates have a <code>.crt</code>, <code>.cer</code> or <code>.pem</code> extension.</p>
<code>-pfx <pfx certificate file></code>	<p>If you have a Personal Information Exchange (PFX) certificate, you can use this parameter to specify its location, instead of using a PEM certificate and key. Typically, such certificates have a <code>.pfx</code> or <code>.p12</code> extension. Since these certificates include their own private key, specifying a key with this certificate type is not required.</p>
<code>-ca <root ca certificate file></code>	<p>If you have a domain certificate that is signed by a certificate authority (CA), this parameter allows you to specify the root CA certificate and/or its intermediate certificates.</p> <p>If you want to specify several certificates so as to compose the chain going from your domain certificate to the root CA certificate, you can repeat the parameter, like so:</p> <pre style="text-align: center;">-ca <intermediate certificate 1> -ca <intermediate certificate 2> -ca <root CA certificate></pre> <p>Typically, you will be able to log on to WebCC using SSL from a Web browser even if WebCC is only configured with your domain certificate, provided that it is signed by a reputable certificate authority. However, if you want project collaborators to be able to use the Download Database Changes and Update Database Scene functionalities in Harmony via SSL, it may be necessary to specify the root CA certificates and/or its intermediates, as Harmony may not otherwise be able to recognize the authority of the CA who signed your certificate.</p> <p>Typically, you can obtain the root CA certificates or intermediate certificates from the root certificate authority of the certificate authority who signed your domain certificate.</p> <div data-bbox="565 1499 1430 1709" style="border: 1px solid green; padding: 10px; margin-top: 20px;"> <p> NOTE</p> <p>If necessary, you can allow Harmony to trust the WebCC server even if it has a self-signed certificate—see Disabling SSL Certificate Validation for WebCC in Harmony on page 280.</p> </div>

Setting Up Static IPs for WebCC on Windows

It is common to configure server machines to have a static IP. This avoids having to rely on DNS resolution to find the server when its IP address changes, and it also allows you to associate the hostname to the server's static IP address on the machine to skip DNS resolution altogether, increasing network performance.

If your machine is currently connected to a DHCP server that dynamically assigns IP addresses, you can configure your machine to permanently use its current IP address. To do this, you can start by making note of the information for your current network connection, then using these parameters for a permanent IP address.

How to obtain the information about your current connection

1. Open Command Prompt and enter the following command:

```
> ipconfig /all
```

This will output information about your current connection:

```
Windows IP Configuration

Host Name . . . . . : <hostname>
Primary Dns Suffix . . . . . : <domain name>
Node Type . . . . . : Peer-Peer
IP Routing Enabled. . . . . : No
WINS Proxy Enabled. . . . . : No
DNS Suffix Search List. . . . . : <domaine names>

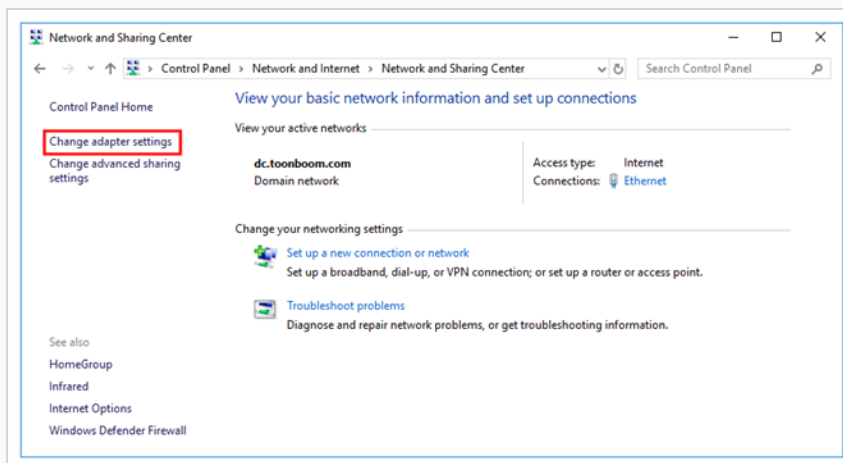
Ethernet adapter Ethernet:

Connection-specific DNS Suffix . : <domain name>
Description . . . . . : <adapter name>
Physical Address. . . . . : <MAC address>
DHCP Enabled. . . . . : Yes
Autoconfiguration Enabled . . . . : Yes
Link-local IPv6 Address . . . . . : <IPv6 address>
IPv4 Address. . . . . : <IPv4 address>
Subnet Mask . . . . . : <subnet mask>
Lease Obtained. . . . . : <date of lease>
Lease Expires . . . . . : <date of expiration>
Default Gateway . . . . . : <default gateway>
DHCP Server . . . . . : <DHCP server address>
DHCPv6 IAID . . . . . : <DHCPv6 IAID>
DHCPv6 Client DUID. . . . . : <DHCPv6 Client DUID>
DNS Servers . . . . . : <DNS server addresses>
Primary WINS Server . . . . . : <WINS server address>
NetBIOS over Tcpip. . . . . : Enabled
```

2. In the output, under Ethernet adapter Ethernet, make note of the following data:
 - The domains in the **DNS Suffix Search List**
 - The **IPv4 Address**
 - The **Subnet Mask**
 - The **Default Gateway** address
 - The addresses for the **DNS Servers**

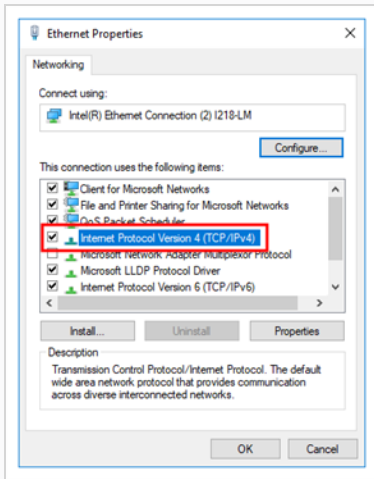
How to configure a Windows server to use a static IP

1. Open the Windows Control Panel:
 - On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
 - On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.
2. Click **View network status and tasks**.
3. On the top-left corner of the Network and Sharing Center, select **Change adapter settings**.



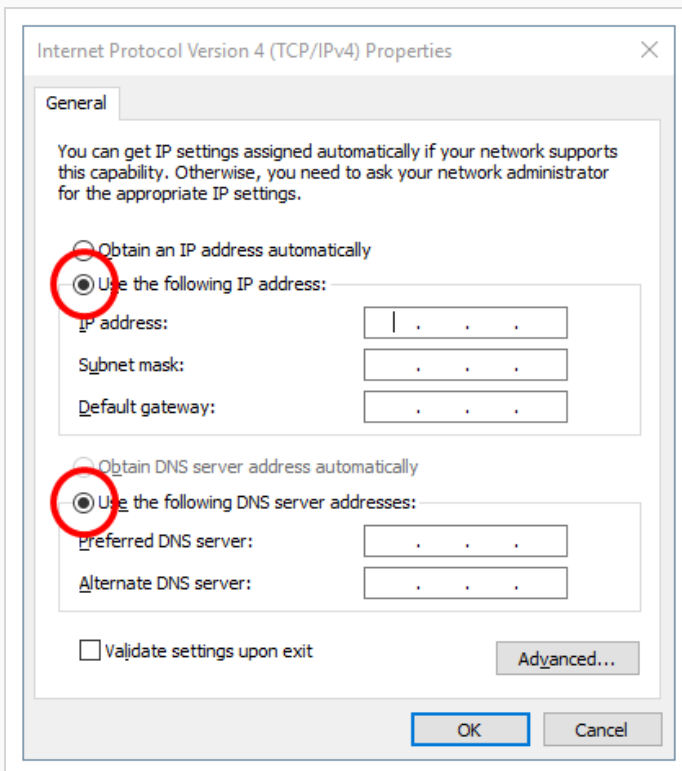
4. Right-click on the network adapter used by your machine to connect to your local network, then select **Properties**.

The Local Area Connection Properties dialog appears.



5. In the items list, select **Internet Protocol Version 4 (TCP/IPv4)**.
6. Click on the **Properties** button under the list.

The Internet Protocol Version 4 (TCP/IPv4) Properties dialog box opens.

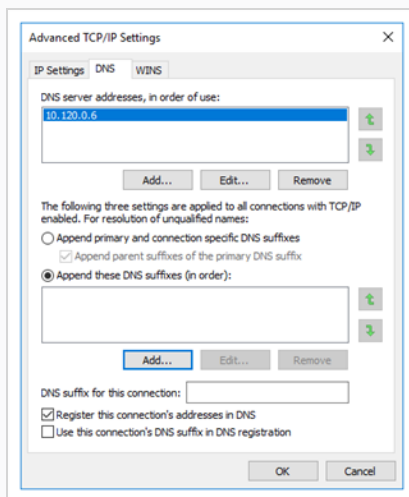


7. Select the **Use the following IP address** option.
8. In the **IP Address** field, enter the **IPv4 Address** you obtained in earlier.

9. In the **Subnet mask** field, enter the **Subnet mask** you obtained earlier.
10. In the **Default gateway** field, enter the **Default gateway** address you obtained earlier.
11. Select the **Use the following DNS server addresses** option.
12. In the **Preferred DNS server** field, enter the **DNS Servers** address obtained earlier.
13. If a second DNS server was listed by ipconfig, type it in the **Alternate DNS server** field. Otherwise, leave this field empty.
14. Click on **Advanced**.

The Advanced TCP/IP Settings dialog opens.

15. Select the **DNS** tab.



16. Select the **Append these DNS suffixes (in order):** option.
17. For each DNS Suffix obtained earlier, click on **Add**, then type in the suffix. Add the suffixes in the order in which there were listed.
18. Click on **OK**.
19. Click on **OK** in the Internet Protocol Version 4 (TCP/IPv4) Properties dialog.
20. Click on **OK** in the Ethernet Properties dialog.

Opening the Firewall Ports for a Harmony WebCC Server on Windows

By default, the Windows firewall will block the communication between client workstations attempting to connect to the Harmony server and the database server service. In order to allow clients to connect to your server, you must open the ports used by the database server on the server machine.

How to open the ports for the WebCC server on Windows

1. Open the Windows Control Panel:
 - On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
 - On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.

2. Click on **System and Security**.
3. Click on **Windows Defender Firewall**.
4. Click on **Advanced settings**.

The Windows Defender Firewall with Advanced Security window opens.

5. In the list to the left, select **Inbound Rules**.
6. In the **Actions** list to the right, select **New Rule**.

The New Inbound Rule Wizard appears.

7. Under **What type of rule would you like to create?**, select **Port** and click on **Next**.
8. Under **Does this rule apply to TCP or UDP?**, select **TCP**.
9. Under **Does this rule apply to all local ports or specific local ports?**, select **Specify local ports**.
10. In the text field, type the port number **8080**.



NOTE

8080 is the default port used by the WebCC server. You can customize this port number—see [Configuring WebCC with Custom Parameters on Windows on page 201](#).

11. Click on **Next**.
12. Under **What action should be taken [...]**, select **Allow the connection** and click on **Next**.
13. Under **When does this rule apply?**, check the network types over which clients will attempt to connect to this server, then click on **Next**.

14. In the name field, name the rule "Toon Boom WebCC Server".

15. Click on **Finish**.

Clients on your network will now be able to connect to the WebCC server.



NOTE

If your WebCC server is configured to allow both http and https connections, you will need to add a firewall rule for the port used by both protocols.


Checking the WebCC Server Log on Windows

In order to troubleshoot problems with a WebCC server, you can look through its log. This is done differently depending on whether you installed the database server as a Windows service using Configuration Wizard or using Service Launcher.

How to check the log for the WebCC server service

1. Open an Explorer window.
2. Browse to the following location:
`C:\ProgramData\Toon Boom Animation\Toon Boom Harmony`
3. Right-click on `WebControlCenter.log` and select **Open with > Notepad**.

How to check the log for the WebCC server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the notification tray, click on the  Service Launcher icon.
The Service Launcher application opens.
3. Select the **WebCC** tab.
4. In the bottom half, select the **Log** tab.
5. Click on the **View Complete Log** button in the bottom-right corner.

Launching the WebCC Server Manually on Windows

You can start the WebCC server directly from a command line. This is usually not practical for a production environment, but it can be very useful when troubleshooting problems. You can view the output of the server directly in the command line as it launches and handles requests.

How to start the WebCC server manually


1. In a command line, enter the following command:

```
> C:\Program Files (x86)\Toon Boom Animation\Toon Boom Harmony 17  
[Edition]\win64\bin\webcc.bat
```

Stopping and Restarting the WebCC Server on Windows

If you installed the WebCC server as a Windows service using Configuration Wizard, you can stop it and restart it using the Harmony Control Panel or the Windows Services control panel. If you installed it using Service Launcher, you can stop it and restart it manually in Service Launcher.

How to stop the WebCC server service using the Harmony Control Panel

1. Open the **Start** menu and, in the **Programs** menu, select **Harmony 17 [Edition] > Tools >  Control Panel**.
2. When prompted, allow Control Panel to make changes to your device.
3. In the **Process Starter** drop-down, select **HarmonyStarter0-webcc.bat**.



NOTE

The number after **HarmonyStarter** may be different, depending on how many Harmony services were installed before this one.

4. Click on **Stop**.
5. When you are ready to start the server again, click on **Start**.

How to stop the WebCC server service using the Windows Services control panel

1. Open the Windows Control Panel:
 - On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
 - On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.
2. In the Control Panel, navigate to **System and Security > Administrative Tools**.
3. Double-click on **Services**.
4. Scroll-down to **HarmonyStarter0-webcc.bat**.




NOTE

The number after **HarmonyStarter** may be different, depending on how many Harmony services were installed before this one.

5. Right-click on the service and select **Stop**.
6. When you are ready to start the server again, right-click on it and select **Start**.


How to stop and restart the WebCC server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the notification tray, click on the  Service Launcher icon.
The Service Launcher application opens.
3. Select the **WebCC** tab.
4. Click on **Stop**.
5. When you are ready to start the server again, click on **Launch**.


Uninstalling the WebCC Server on Windows

If you installed the WebCC server as a Windows service using Configuration Wizard, you can uninstall it using the Configuration Wizard, the Harmony Control Panel or the Windows Services control panel. If you installed it using Service Launcher, you can uninstall it in Service Launcher.

How to uninstall the WebCC service using Configuration Wizard

1. In the **Start** menu, open the **Programs** menu and select **Harmony 17 [Edition] > Tools >  Configuration Wizard**.
2. When prompted, enable Configuration Wizard to make changes to your device.
The Toon Boom Harmony Configuration Wizard launches.
3. In the list of tasks, right of **Harmony Web Control Center Server**, click on
4. In the ensuing dialog, under **Do you wish to install the Harmony Web Control Center Server?**, select **No**.
5. Click on **Next**.
6. Click on **Finish** to confirm the changes.
The WebCC server service will stop and will no longer launch on boot.

How to uninstall the WebCC server service using the Harmony Control Panel

1. Open the **Start** menu and, in the **Programs** menu, select **Harmony 17 [Edition] > Tools >  Control Panel**.
2. When prompted, allow Control Panel to make changes to your device.
3. In the **Process Starter** drop-down, select **HarmonyStarter0-webcc.bat**.



NOTE

The number after **HarmonyStarter** may be different, depending on how many Harmony services were installed before this one.

4. Click on **Stop**.
5. Click on **Remove**.
The WebCC server service will stop and will no longer launch on boot.
6. Close the **Control Panel**.

How to uninstall the WebCC server service using the Windows Services control panel

1. Open the Windows Control Panel:
 - On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
 - On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.
2. In the Control Panel, navigate to **System and Security > Administrative Tools**.
3. Double-click on **Services**.
4. Scroll-down to **HarmonyStarter0-webcc.bat**.




NOTE

The number after **HarmonyStarter** may be different, depending on how many Harmony services were installed before this one.

5. Double-click on the service to open its Properties dialog.
6. In the **Startup type** drop-down, select **Disabled**.
From that point on, the WebCC server service will no longer launch on boot.
7. Click on **OK**.

How to uninstall the WebCC server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the notification tray, click on the  Service Launcher icon.
The Service Launcher application opens.
3. Select the **WebCC** tab.
4. Click on **Stop**.
5. Uncheck the **Launch when Service Launcher starts** option.
6. In the top menu, select **Server > Save All Settings**.

Installing WebCC on macOS

On Windows, installing a WebCC server with the default parameters can easily be done using the Configuration Assistant.

Installing the WebCC Server Daemon on macOS	224
Running the WebCC Server on Log-In on macOS	226
Configuring WebCC with Custom Parameters on macOS	228
About SSL, HTTPS and WebCC on macOS	233
Configuring WebCC for SSL Support on macOS	234
Configuring a macOS WebCC Server to have a Static IP Address	238
Looking Up the WebCCServer Log on macOS	241
Launching the WebCC Server Manually on macOS	242
Stopping and Restarting the WebCC Server on macOS	243
Uninstalling the WebCC Server on macOS	244

Installing the WebCC Server Daemon on macOS

You can easily install WebCC as a daemon using the Configuration Wizard. This daemon runs through the user `usabatch`, so this user must be created prior to installing the daemon.




NOTE

The following steps indicate how to make the WebCC server daemon launch on boot, via the user `usabatch`, who will have administrator privileges. If you prefer, you can make the database server launch on log-in as the logged-in user. For more information, see [Running the Database Server on Log-In on macOS on page 81](#).

How to create the user `usabatch` on macOS

1. Open the Apple menu and select **System Preferences**.
2. Click on **Users and Groups**.
3. In the bottom-left corner of the Users & Groups dialog, click on the lock icon labeled **Click the lock to make changes** to get administrator permissions.
4. Enter the username and password of an account with administrator privileges.
5. Click on the + button at the bottom of the user list.
The user creation panel appears.
6. In the **New Account** drop-down, select **Administrator**.
7. In the **Full Name** and **Account Name** fields, enter `usabatch`.
8. Give the user a password.
9. Click on **Create User**.

How to install the WebCC server daemon on macOS

1. In Finder, proceed to **Applications >  Toon Boom Harmony 17 [Edition]** and double-click **Harmony [Edition]**.
The main Harmony application launches.
2. In the top menu, select **Harmony [Edition] > Tools > Configuration Assistant**.
 1. In the Welcome screen, check the **Create the Startup Items** option and uncheck all other options.
 2. Click on **Continue**.

3. Check the **Create Harmony Web Control Center Server Startup Item** option and uncheck all other options.
4. Click on **Create**.
5. When prompted, enter the username and the password of a user with administrator privileges.

The WebCC daemon is now installed and should be running. You can test that it works by launching a web browser and going to the following address:

`http://localhost:8080/`

Running the WebCC Server on Log-In on macOS

The traditional way of running a WebCC server is by running the server as a daemon. This means it launches and allows clients to connect as soon as the machine is booted, without any user being signed-in.

Using Service Launcher, it is possible to configure your WebCC server to only launch as the currently logged-in user, when they log in. This means that WebCC will only be accessible if the user configured to run it is logged in.

Although this may seem impractical, it may be preferable for security reasons. Services installed with the Configuration Wizard are run under the user **usabatch**, an user with administrator privileges which is created by Configuration Wizard to launch the services. When using Service Launcher, the logged-in user does not need to have administrator privileges to run services.

Before configuring Service Launcher to run the WebCC server, the daemon that launches it must be disabled, so as to avoid launching two instances of the WebCC server simultaneously



NOTE

While daemons run regardless of who is logged in and even when no user is logged-in, Service Launcher will only run when the user for which it was specifically configured is logged in. This means that if you want Service Launcher to launch when any user is logged in, you must configure it for each user that has access to the machine.

How to disable the WebCC server daemon

1. In a terminal, type in the following command:

```
$ sudo launchctl unload
/Library/LaunchDaemons/com.toonboom.WebCCServer.plist
```

The WebCC server daemon will stop and will no longer launch on boot.

How to configure the WebCC server to launch on user log-in

1. Make sure you are currently signed-in as the user who will run the service.




NOTE

This user will need to be logged in for the service to be running. The user does not need to have administrator privileges.

2. Do one of the following:

- If Service Launcher is not running, follow these steps to launch it:
 - a. In Finder, proceed to **Applications > H Toon Boom Harmony 17 [Edition]** and double-click **Harmony [Edition]**.

The main Harmony application launches.
 - b. In the top menu, select **Harmony [Edition] > Tools > Service Launcher**.
- If Service Launcher is already running, click on its icon  in the menu bar.

The Service Launcher application window appears.

3. In the **Application parameters** section of the **Preferences** tab, make sure that the **Launch on login** option is checked, so that services configured in Service Launcher automatically launch when you log in to your account.
4. Select the **WebCC** tab.
5. Check the **Launch when Service Launcher starts** option.
6. To make sure users must enter their password when logging in to WebCC, check the **Authentication** option.
7. By default, WebCC will use port **8080**. If you want to use a specific port to connect to WebCC, type it in the **Port** field.
8. Click on **Launch** to verify that WebCC launches successfully through Service Launcher using the selected parameters.
9. In the top menu, select **Server > Save All Settings**.

From now on, the database server will launch through Service Launcher when you log in



NOTE

You can enter parameters in the Other Parameters field to customize the WebCC server. For more information, see [Configuring WebCC with Custom Parameters on macOS on page 228](#).

Configuring WebCC with Custom Parameters on macOS

WebCC can run with no parameters, but it also provides some customization options. Among other things, you can change the port number, adjust its security settings and configure it to run with SSL security.

If you installed WebCC as a daemon, you can customize its parameters by modifying the daemon configuration file that launches WebCC. If you use Service Launcher to run WebCC, you can easily add parameters in the Other Parameters field of the WebCC tab of Service Launcher.

How to add parameters to the WebCC daemon

1. In a terminal, type in the following command to edit the WebCC daemon:

```
$ sudo vi /Library/LaunchDaemons/com.toonboom.WebCCServer.plist
```

2. Scroll all the way down to the following line:

```
<string>export PATH="$USADIR/macosx/bin:$PATH"; "$USADIR/macosx/bin/webcc.sh"
</string>
```

3. Right after "**\$USADIR/macosx/bin/webcc.sh**", add a space, then type each parameter you want to add, separated by spaces. For example, if you want WebCC to use the port number **8081**, you would edit the line like so:

```
<string>export PATH="$USADIR/macosx/bin:$PATH"; "$USADIR/macosx/bin/webcc.sh" -
port 8081</string>
```




NOTE

For a list of parameters and their usage, see [WebCC Parameters Table on page 229](#).

4. Save and quit.
5. Restart the WebCC daemon:

```
$ sudo launchctl load
/Library/LaunchDaemons/com.toonboom.WebCCServer.plist
```

How to add parameters for WebCC in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the top menu, click on the Service Launcher  icon.

The Service Launcher application opens.

3. Double-click on **Service Launcher**.

The Service Launcher application launches.

4. Select the **WebCC** tab.
5. If you want to let any user connect to WebCC without credentials and have administrative privileges, uncheck the **Authentication** option. Otherwise, make sure it is checked.
6. If you want to change the port used by the WebCC server, type the desired port number in the **Port** field.
7. If you want to specify any other option, type them in the **Other Parameters** field, separated by spaces.

**NOTE**


For a list of parameters and their usage, see [WebCC Parameters Table on page 229](#).

8. In the top menu, select **Server > Save All Settings**.
9. Click on the **Stop** button.
10. Click on the **Launch** button.

WebCC Parameters Table

The following is a table of parameters that can be specified to customize the behaviour of your WebCC server. They can be used in both the WebCC shell script and in Service Launcher.

Parameter	Description
<code>-numWorkers <number></code>	<p>The amount of server nodes to launch to process client requests. Each node requires one CPU core, and a single node can only process one client request at a time. Therefore, the more nodes are launched, the less likely the server is to stall under heavy load.</p> <p>If this parameter is not specified, WebCC will use as many nodes as there are CPU cores on the server.</p>
<code>-noAuth</code>	<p>Disables authentication. With this option enabled, any user accessing the WebCC web interface will be logged in as usabatch and will have administrative privileges on the entire database.</p>
<code>-port <port number></code>	<p>The port to which WebCC is deployed.</p> <p>The default port for accessing WebCC is 8080. For browsers to access WebCC</p>

Parameter	Description
	<p>without specifying the port number, you can use port 80.</p> <div style="border: 1px solid green; padding: 5px; margin-top: 10px;">  NOTE If either the -http or -https parameter is specified, the -port parameter will be ignored. </div>
-exchange <directory>	<p>The location of the exchange directory on the local filesystem. This directory is where all packages are stored when exporting or importing scenes.</p> <p>By default, this directory is /USA_DB/exchange.</p>
-cacheDuration <seconds>	<p>The amount of seconds to keep data in the cache. This can be set to any value between 1 and 999999. By default, this is set to 3600.</p>
-allowAllOrigin	<p>Enables cross-origin resource sharing on the WebCC server.</p> <p>By default, as a standard security measure, only files on the same server as the WebCC server are allowed to use WebCC resources. With this option enabled, it is possible to create web interfaces on other servers that use resources stored on the WebCC server, or to store different modules of WebCC on different servers and let them communicate together.</p> <p>In most use cases, this option should be disabled to prevent other machines from exploiting resources on the WebCC server.</p>
-disableAnalytics	<p>Disables Google Analytics tracking for all clients. By default, this is enabled.</p>
-permission <user type> <action> <on off>	<p>Overrides a default permission for the given user type.</p> <p>At the moment, this parameter can only be used for the user type animator and for the permission import_export_any_scene. Hence, when used with the following syntax, animator-type users can export and import any scene in the database, without requiring the scene to be assigned to them first.</p> <p style="text-align: center;">-permission animator import_export_any_scene on</p>


HTTPS/SSL Parameters Table

The following parameters can be used to enable and configure SSL for your WebCC server.

**NOTE**

For more information on how to configure WebCC with SSL, see [About SSL, HTTPS and WebCC on macOS](#) on page 233.

Parameter	Description
<code>-https <port number></code>	<p>Enables connection to the server through the https protocol using the specified port number.</p> <p>A commonly used port for accessing WebCC through the https protocol is 8443. For web browsers to access WebCC via https without specifying the port number—provided that the address of the server is preceded by https://—you can use port 443.</p> <p>For https to work, you must either specify a pair of PEM format key and certificate with the -key and -cert option, or a PFX format certificate with the -pfx option.</p>
<code>-http <port number></code>	<p>If access through the https protocol is enabled, this parameter allows you to also allow access through the http protocol through the specified port.</p> <p>Using this parameter is optional. If the -https parameter is specified, but not the -http parameter, WebCC will only accept connections through https.</p> <p>The default port for accessing WebCC through the http protocol is 8080. For browsers to access WebCC via http without specifying the port number, you can use port 80.</p>
<code>-key <key file></code>	<p>The location of the private key file for the SSL session. This must be specified along with the -cert parameter.</p> <p>The private key must be in Privacy Enhanced Mail (PEM) format, which is an ASCII format where the key is encoded in Base64 and located between -----BEGIN PRIVATE KEY----- and -----END PRIVATE KEY----- tags. Typically private keys in this format have a .key extension.</p>
<code>-cert <certificate file></code>	<p>The location of the security certificate file for the SSL session. This must be specified along with the -key parameter.</p> <p>The certificate must be in Privacy Enhanced Mail (PEM) format, which is an ASCII format where the certificate is encoded in Base64 and located between -----BEGIN CERTIFICATE----- and -----END CERTIFICATE----- tags. Typically, such certificates have a .cert, .cer or .pem extension.</p>
<code>-pfx <pfx certificate file></code>	<p>If you have a Personal Information Exchange (PFX) certificate, you can use this parameter to specify its location, instead of using a PEM certificate and key.</p>

Parameter	Description
	<p>Typically, such certificates have a .pfx or .p12 extension. Since these certificates include their own private key, specifying a key with this certificate type is not required.</p>
<p>-ca <root ca certificate file></p>	<p>If you have a domain certificate that is signed by a certificate authority (CA), this parameter allows you to specify the root CA certificate and/or its intermediate certificates.</p> <p>If you want to specify several certificates so as to compose the chain going from your domain certificate to the root CA certificate, you can repeat the parameter, like so:</p> <pre style="text-align: center;">-ca <intermediate certificate 1> -ca <intermediate certificate 2> -ca <root CA certificate></pre> <p>Typically, you will be able to log on to WebCC using SSL from a Web browser even if WebCC is only configured with your domain certificate, provided that it is signed by a reputable certificate authority. However, if you want project collaborators to be able to use the Download Database Changes and Update Database Scene functionalities in Harmony via SSL, it may be necessary to specify the root CA certificates and/or its intermediates, as Harmony may not otherwise be able to recognize the authority of the CA who signed your certificate.</p> <p>Typically, you can obtain the root CA certificates or intermediate certificates from the root certificate authority of the certificate authority who signed your domain certificate.</p> <div style="border: 1px solid green; padding: 10px; margin-top: 10px;"> <p> NOTE</p> <p>If necessary, you can allow Harmony to trust the WebCC server even if it has a self-signed certificate—see Disabling SSL Certificate Validation for WebCC in Harmony on page 280.</p> </div>

About SSL, HTTPS and WebCC on macOS

You can configure your WebCC server to accept connections through the https protocol, either exclusively or alongside the http protocol. Connections through https are encrypted and the identity of the server is verified using a certificate. This ensures the information exchanged between the client and the server cannot be deciphered if intercepted. Also, provided that you configure WebCC with a certificate signed by a reputable certificate authority, this ensures that hackers cannot perform man-in-the-middle attacks by making their own server pose as your WebCC server to intercept its traffic.

For WebCC to accept connections with the https protocol, you must configure WebCC with the following:

- The port used for https connections.
- The private certificate key.
- The certificate.
- Optionally, the root CA and/or intermediate certificate.

There are two types of certificates supported by WebCC:

- **Privacy Enhanced Mail (PEM) certificates:** These certificates are ASCII files encoded in Base64. Typically, the certificate is in a file with a `.crt`, `.cer` or `.pem` extension and the private key is in a file with a `.key` extension.
- **Personal Information Exchange (PFX /PKCS #12) certificates:** These certificates are binary files that include both the certificate and its private key. Typically, such certificates are used for Windows servers and have a `.pfx` or `.p12` extension.

If your certificate is in any other format, such as DER, PKCS #7 or CA_bundle, you will need to perform extra steps to convert the certificate to a supported format.

About Certificate Signature

It is recommended to use a certificate that has been signed by a reputable certificate authority.

If your certificate is signed by a certificate authority, WebCC will typically be accessible without issue in a web browser. However, if you want clients to be able to upload and download scene changes directly between Harmony and the WebCC server, you may have to specify the root and/or intermediate certificate for your certificate. This is because web browsers are typically designed to trace a certificate's chain of trust all the way to the root certificate authority, whereas Harmony is not.

If your certificate is self-signed, web browsers will warn the user that the website is not secure, and users will need to go around that warning to access WebCC. Likewise, Harmony will not be able to upload or download scene changes directly from its interface at all, unless the client makes Harmony trust self-signed certificates by enabling the hidden preference `WEBCC_SSL_SELF_SIGNED`—see [Disabling SSL Certificate Validation for WebCC in Harmony on page 280](#).

Configuring WebCC for SSL Support on macOS

Like other WebCC options, configuring WebCC to use SSL is done by using command line parameters when launching WebCC. To use WebCC with SSL, you must specify the `-https` parameter followed by the port WebCC should use for https connections, the certificate and key file.

The following are examples of parameters to use for configuring WebCC to use SSL:

- If you have a PEM certificate and key file:

```
-https 8443 -key <path to PEM key file> -cert <path to PEM certificate>
```

- If you have a PEM certificate and its private key in the same file, specify the same file using both the `-key` and `-cert` parameters:

```
-https 8443 -key <path to PEM certificate> -cert <path to PEM certificate>
```

- If you have a PFX certificate, simply specify its path with the `-pfx` parameter:

```
-https 8443 -pfx <path to PFX certificate>
```

- If you want to allow users to connect to your server via either http or https, you can do so by specifying a port number for each protocol:

```
-http 8080 -https 8443 -key <path to key file> -cert <path to certificate>
```

- If you have a root certificate, an intermediate certificate, or both, you can specify each file preceded by the `-ca` parameter, in any order:

```
-http 8443 -key <path to key file> -cert <path to certificate> -ca <path to intermediate certificate 1> -ca <path to intermediate certificate 2> -ca <path to root certificate>
```

How to add parameters to the WebCC daemon

1. In a terminal, type in the following command to edit the WebCC daemon:

```
$ sudo vi /Library/LaunchDaemons/com.toonboom.WebCCServer.plist
```

2. Scroll all the way down to the following line:

```
<string>export PATH="$USADIR/macosx/bin:$PATH"; "$USADIR/macosx/bin/webcc.sh"
</string>
```

3. Right after "`$USADIR/macosx/bin/webcc.sh`", add a space, then type each parameter you want to add, separated by spaces. For example, if you want WebCC to use the port number **8081**, you would edit the line like so:

```
<string>export PATH="$USADIR/macosx/bin:$PATH"; "$USADIR/macosx/bin/webcc.sh" -  
port 8081</string>
```


**NOTE**

For a list of parameters and their usage, see [HTTPS/SSL Parameters Table on page 236](#).

4. Save and quit.
5. Restart the WebCC daemon:

```
$ sudo launchctl load  
/Library/LaunchDaemons/com.toonboom.WebCCServer.plist
```

How to add parameters for WebCC in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the top menu, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Double-click on **Service Launcher**.
The Service Launcher application launches.
4. Select the **WebCC** tab.
5. If you want to let any user connect to WebCC without credentials and have administrative privileges, uncheck the **Authentication** option. Otherwise, make sure it is checked.
6. If you want to change the port used by the WebCC server, type the desired port number in the **Port** field.
7. If you want to specify any other option, type them in the **Other Parameters** field, separated by spaces.

**NOTE**


For a list of parameters and their usage, see [HTTPS/SSL Parameters Table on page 236](#).

8. In the top menu, select **Server > Save All Settings**.
9. Click on the **Stop** button.
10. Click on the **Launch** button.

HTTPS/SSL Parameters Table

The following parameters can be used to enable and configure SSL for your WebCC server.

Parameter	Description
<code>-https <port number></code>	<p>Enables connection to the server through the https protocol using the specified port number.</p> <p>A commonly used port for accessing WebCC through the https protocol is 8443. For web browsers to access WebCC via https without specifying the port number—provided that the address of the server is preceded by https://—you can use port 443.</p> <p>For https to work, you must either specify a pair of PEM format key and certificate with the -key and -cert option, or a PFX format certificate with the -pfx option.</p>
<code>-http <port number></code>	<p>If access through the https protocol is enabled, this parameter allows you to also allow access through the http protocol through the specified port.</p> <p>Using this parameter is optional. If the -https parameter is specified, but not the -http parameter, WebCC will only accept connections through https.</p> <p>The default port for accessing WebCC through the http protocol is 8080. For browsers to access WebCC via http without specifying the port number, you can use port 80.</p>
<code>-key <key file></code>	<p>The location of the private key file for the SSL session. This must be specified along with the -cert parameter.</p> <p>The private key must be in Privacy Enhanced Mail (PEM) format, which is an ASCII format where the key is encoded in Base64 and located between -----BEGIN PRIVATE KEY----- and -----END PRIVATE KEY----- tags. Typically private keys in this format have a .key extension.</p>
<code>-cert <certificate file></code>	<p>The location of the security certificate file for the SSL session. This must be specified along with the -key parameter.</p> <p>The certificate must be in Privacy Enhanced Mail (PEM) format, which is an ASCII format where the certificate is encoded in Base64 and located between -----BEGIN CERTIFICATE----- and -----END CERTIFICATE----- tags.</p>

Parameter	Description
	Typically, such certificates have a <code>.crt</code> , <code>.cer</code> or <code>.pem</code> extension.
<pre>-pfx <pfx certificate file></pre>	<p>If you have a Personal Information Exchange (PFX) certificate, you can use this parameter to specify its location, instead of using a PEM certificate and key. Typically, such certificates have a <code>.pfx</code> or <code>.p12</code> extension. Since these certificates include their own private key, specifying a key with this certificate type is not required.</p>
<pre>-ca <root ca certificate file></pre>	<p>If you have a domain certificate that is signed by a certificate authority (CA), this parameter allows you to specify the root CA certificate and/or its intermediate certificates.</p> <p>If you want to specify several certificates so as to compose the chain going from your domain certificate to the root CA certificate, you can repeat the parameter, like so:</p> <pre>-ca <intermediate certificate 1> -ca <intermediate certificate 2> -ca <root CA certificate></pre> <p>Typically, you will be able to log on to WebCC using SSL from a Web browser even if WebCC is only configured with your domain certificate, provided that it is signed by a reputable certificate authority. However, if you want project collaborators to be able to use the Download Database Changes and Update Database Scene functionalities in Harmony via SSL, it may be necessary to specify the root CA certificates and/or its intermediates, as Harmony may not otherwise be able to recognize the authority of the CA who signed your certificate.</p> <p>Typically, you can obtain the root CA certificates or intermediate certificates from the root certificate authority of the certificate authority who signed your domain certificate.</p> <div data-bbox="565 1339 1430 1545" style="border: 1px solid green; padding: 10px; margin-top: 20px;"> <p> NOTE</p> <p>If necessary, you can allow Harmony to trust the WebCC server even if it has a self-signed certificate—see Disabling SSL Certificate Validation for WebCC in Harmony on page 280.</p> </div>

Configuring a macOS WebCC Server to have a Static IP Address

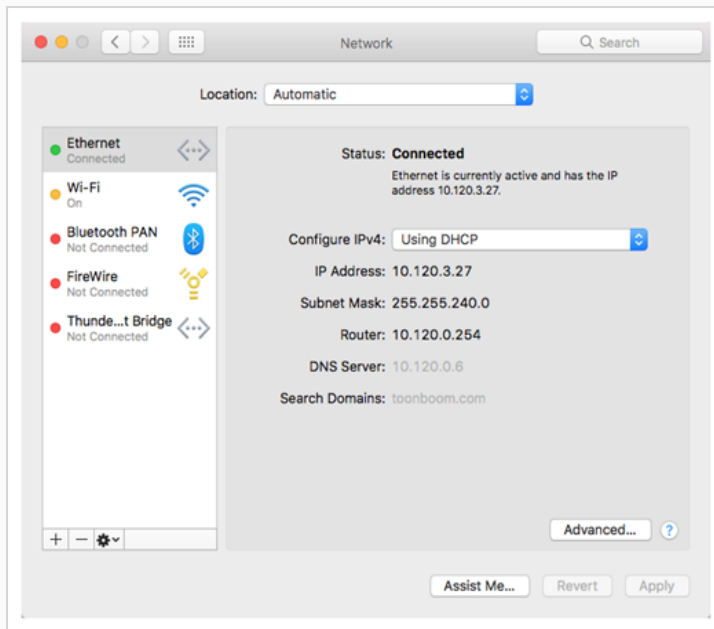
It is common to configure server machines to have a static IP. This avoids having to rely on DNS resolution to find the server when its IP address changes, and it also allows you to associate the hostname to the server's static IP address on the machine to skip DNS resolution altogether, increasing network performance.

If your machine is currently connected to a DHCP server that dynamically assigns IP addresses, you can configure your machine to permanently use its current IP address. To do this, you can start by making note of the information for your current network connection, then using these parameters for a permanent IP address.

How to obtain information about your current connection on macOS

1. Open the Apple menu and select **System Preferences**.
2. Click on **Network**.

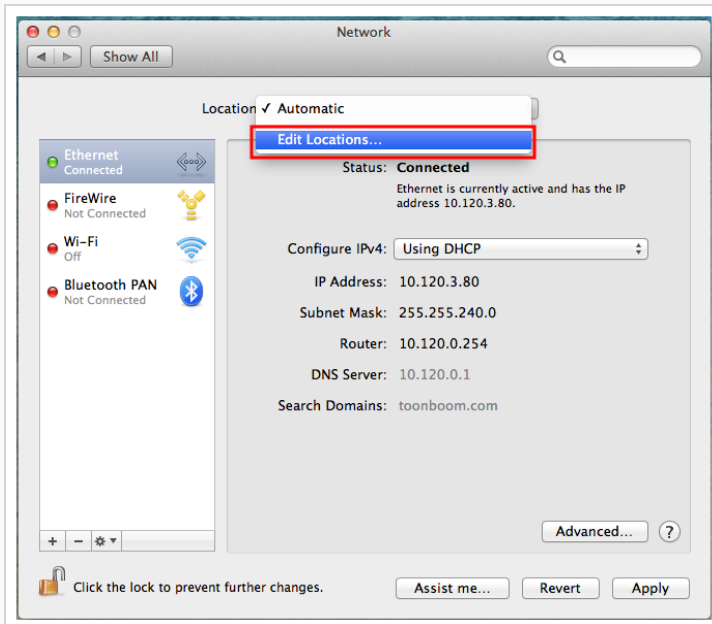
The Network configuration panel appears.



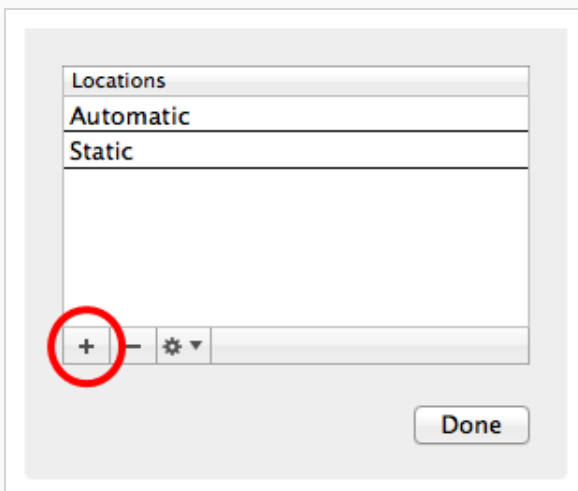
3. In the list to the left, select **Ethernet**.
4. In the panel to the right, make note of the following information:
 - The **IP Address**
 - The **Subnet Mask**
 - The **Router** address
 - The **DNS Server**
 - The **Search Domains**

How to configure a macOS server to use a static IP

1. Open the Apple menu and select **System Preferences**.
2. Click on **Network**.
3. In the list to the left, select **Ethernet**.
4. In the Location drop-down, select Edit Locations.



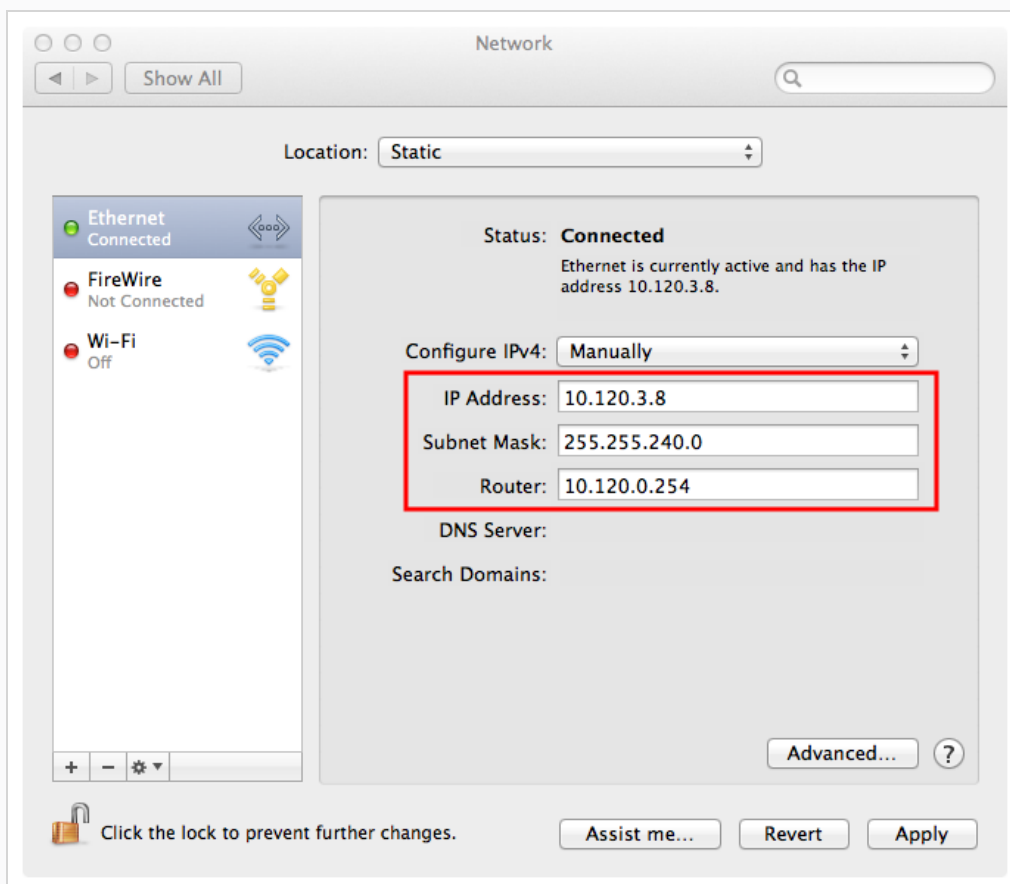
5. Click on the + button to add a location, then name it Static.



6. Click on **Done**.

7. Click on **Advanced**.
8. In the **TCP/IP** tab of the Advanced Network panel, enter the following information:
 - Set the **IPv4 Address** to the **IP Address** obtained earlier.
 - Set the **Subnet Mask** to the **Subnet Mask** obtained earlier.
 - Set the **Router** address to the **Router** address obtained earlier.
9. Open the **DNS** tab.
10. Under the **DNS Servers** list, click on the + button to add a DNS server. Add each of the DNS Server addresses obtained earlier.
11. Under the **Search Domains** list, click on the + button to add a search domain. Add each of the search domains obtained earlier.
12. Click on **OK**.

If your configuration works, the Network panel will indicate **Status: Connected** at the top.



Looking Up the WebCCServer Log on macOS


In order to troubleshoot problems with a WebCC server, you can look through its log. This is done differently depending on whether you installed the database server as a daemon using Configuration Assistant, or using Service Launcher.

How to check the log for the WebCC server daemon

1. In a terminal, enter the following command:

```
$ less /tmp/com.toonboom.WebCCServer.log
```

How to check the log for the WebCC server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the top menu, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **WebCC** tab.
4. In the bottom half, select the **Log** tab.
5. Click on the **View Complete Log** button in the bottom-right corner.

Launching the WebCC Server Manually on macOS

You can start the WebCC server directly from a terminal. This is usually not practical for a production environment, but it can be very useful when troubleshooting problems. You can view the output of the server directly in the command line as it launches and handles requests.

How to start the WebCC server manually

1. In a command line, enter the following command:

```
$ "/Applications/Toon Boom Harmony 17 [Edition]/Harmony  
[Edition].app/Contents/tba/macosx/bin/webcc.sh"
```

Stopping and Restarting the WebCC Server on macOS

If you installed the WebCC server as a daemon using the Configuration Assistant, you can stop it and restart it using the `launchctl` command line tool. If you installed it using Service Launcher, you can stop it and restart it manually in Service Launcher.

How to stop and restart the WebCC server daemon


1. In a terminal, enter the following command:

```
$ sudo launchctl unload  
/Library/LaunchDaemons/com.toonboom.WebCCServer.plist
```

2. When you are ready to start the server again, enter the following command:

```
$ sudo launchctl load  
/Library/LaunchDaemons/com.toonboom.WebCCServer.plist
```

How to stop and restart the WebCC server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the top menu, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **WebCC** tab.
4. Click on **Stop**.
5. When you are ready to start the server again, click on **Launch**.

Uninstalling the WebCC Server on macOS

If you installed the WebCC server as a daemon using the Configuration Assistant, you can uninstall it using the `launchctl` command line tool. If you installed it using Service Launcher, you can uninstall it in Service Launcher.

How to uninstall the WebCC server daemon

1. In a terminal, enter the following command:


```
$ sudo launchctl unload -w  
/Library/LaunchDaemons/com.toonboom.WebCCServer.plist
```

The daemon will no longer launch on boot.

2. If you want, you can delete the daemon permanently with the following command:

```
$ sudo rm /Library/LaunchDaemons/com.toonboom.WebCCServer.plist
```

How to uninstall the WebCC server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the top menu, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **WebCC** tab.
4. Click on **Stop**.
5. Uncheck the **Launch when Service Launcher starts** option.
6. In the top menu, select **Server > Save All Settings**.

Installing WebCC on GNU/Linux

On Windows, installing a WebCC server with the default parameters can easily be done using the Harmony installation script.

Disabling Security for a WebCC Server on GNU/Linux	246
Installing WebCC on GNU/Linux	248
Running the WebCC Server on Log-In on GNU/Linux	250
Configuring WebCC with Custom Parameters on GNU/Linux	252
About SSL, HTTPS and WebCC on GNU/Linux	257
Configuring WebCC for SSL Support on GNU/Linux	258
Setting Up Static IPs for WebCC on GNU/Linux	262
Opening the Firewall Ports for a WebCC Server on GNU/Linux	267
Checking the WebCC Server Log on GNU/Linux	268
Launching the WebCC Server Manually on GNU/Linux	269
Stopping and Restarting the WebCC Server on GNU/Linux	270
Uninstalling the WebCC Server on GNU/Linux	271

Disabling Security for a WebCC Server on GNU/Linux

GNU/Linux comes bundled with two core security components:

- A firewall (typically either **iptables** or **firewalld**), which will block incoming connections that were not specifically allowed.
- **SELinux**, a kernel security module that enforces access control policies on software, files and network resources.

The default configuration for these components will prevent clients from connecting to a WebCC server. This installation guide provides instructions on how to add security exceptions to these components so that they do not have to be disabled. However, you may prefer to disable them so as to simplify the installation process or to rule out security components as possible causes when troubleshooting an issue.



IMPORTANT

If you choose to disable security on GNU/Linux, it is critical to ensure that your domain has effective site-wide security mechanisms to prevent unauthorized access to your WebCC server.

How to disable the firewall on CentOS 6, RedHat Enterprise Linux 6 or other systems using the iptables service

1. In a terminal, enter the following commands:

```
$ sudo chkconfig iptables off
$ sudo service iptables stop
```

How to disable the firewall on CentOS 7, RedHat Enterprise Linux 7 or other systems using the firewalld service

1. In a terminal, enter the following commands:

```
$ sudo systemctl disable firewalld
$ sudo systemctl stop firewalld
```

How to disable SELinux

1. In a terminal, enter the following command:

```
$ sudo vi /etc/selinux/config
```

2. Find the line with the following parameter:

```
SELINUX=enforcing
```

3. Replace **enforcing** with **disabled**:

```
SELINUX=disabled
```



IMPORTANT

Double-check to make sure you did not make any typo. An error in the SELinux configuration file may cause GNU/Linux to fail to boot.

4. Save and quit.
5. Reboot to apply the new configuration.

Installing WebCC on GNU/Linux

The Harmony installer for GNU/Linux always installs daemon scripts on your machine, but does not enable them by default. Hence, there are two approaches to installing the WebCC server daemon:

- By enabling the daemon that was already installed with Harmony using the **chkconfig** command.
- By running the Harmony installation script with the **-w** parameter.

The installation of a WebCC server also requires copying a script from the Harmony installation package. The installer does this automatically when it is launched with the **-w** parameter. If you choose to enable the WebCC daemon manually, you will also have to copy this script manually.

This daemon runs through the user `usabatch`, so this user must be created prior to installing the daemon.



NOTE

The following steps indicate how to make the WebCC server daemon launch on boot, via the user `usabatch`. If you prefer, you can make the database server launch on log-in as the logged-in user. For more information, see [Running the Database Server on Log-In on macOS on page 81](#).

How to create the `usabatch` user on GNU/Linux

1. In a terminal, enter the following command:

```
$ sudo useradd -m usabatch
$ sudo passwd usabatch
```

2. When prompted, enter the password you want to assign to the user `usabatch`, then confirm it.

The user `usabatch` is now created and can be used to run Harmony daemons.

How to enable the WebCC server daemon on GNU/Linux

1. In a terminal, enter the following command to check if the rendering script for WebCC is already in your database:

```
$ stat /USA_DB/scripts/TB_WebCC_Render.js
```

2. If the script is not in your database, copy it from the Harmony installation directory:

```
$ mkdir -p -m 777 /USA_DB/scripts
$ cp /usr/local/ToonBoomAnimation/harmony[Edition]_
17/resources/scripts/TB_WebCC_Render.js /USA_DB/scripts/
```

3. In a terminal, enter the following command to enable the WebCC server daemon:

```
$ sudo chkconfig toonboom_webcc on
```

If the command returns nothing, it succeeded. The WebCC server will launch next time you boot the machine.

4. To launch the WebCC server daemon immediately, run the following command:

```
$ sudo service toonboom_webcc start
```

The command will report whether it succeeded or failed.

How to enable the WebCC server daemon using the Harmony installation script

1. In a terminal, navigate to the directory where you downloaded the installation package.

```
$ cd ~/Downloads
```

2. Extract the package's contents in the directory.

```
$ tar -xvzf ./name-of-package.tar.gz
```

3. Switch to the directory in which the package's contents were extracted.

```
$ cd name-of-package
```

4. Run the installer with the `-d` parameter:

```
$ sudo ./install -w
```

5. When prompted, confirm that you want to proceed with the installation.

The installation script will enable and start the WebCC server daemon.

Running the WebCC Server on Log-In on GNU/Linux

The traditional way of running a WebCC server is by running the server as a daemon. This means it launches and allows clients to connect as soon as the machine is booted, without any user being signed-in.

Using Service Launcher, it is possible to configure your WebCC server to only launch as the currently logged-in user, when they log in. This means that WebCC will only be accessible if the user configured to run it is logged in.

Although this may seem impractical, it may be preferable for security reasons. Services installed with the Configuration Wizard are run under the user **usabatch**, an user with administrator privileges which is created by Configuration Wizard to launch the services. When using Service Launcher, the logged-in user does not need to have administrator privileges to run services.

Before configuring Service Launcher to run the WebCC server, the daemon that launches it must be disabled, so as to avoid launching two instances of the WebCC server simultaneously



NOTE

While daemons run regardless of who is logged in and even when no user is logged-in, Service Launcher will only run when the user for which it was specifically configured is logged in. This means that if you want Service Launcher to launch when any user is logged in, you must configure it for each user that has access to the machine.

How to disable the WebCC server daemon

1. In a terminal, type in the following commands to stop, then disable the daemon:

```
$ sudo service toonboom_webcc stop
$ sudo chkconfig toonboom_webcc off
```

The WebCC server daemon will stop and will no longer launch on boot.

How to configure the WebCC server to launch on user log-in



1. Make sure you are currently signed-in as the user who will run the service.



NOTE

This user will need to be logged in for the service to be running. The user does not need to have administrator privileges.

2. Do one of the following:

- If Service Launcher is not running, in the GNOME top menu, select **Applications > Harmony 17 [Edition] >  Service Launcher**.
- If Service Launcher is already running, click on its icon  in the system tray.

The Service Launcher application starts.

3. In the **Application parameters** section of the **Preferences** tab, make sure that the **Launch on login** option is checked, so that services configured in Service Launcher automatically launch when you log in to your account.
4. Select the **WebCC** tab.
5. Check the **Launch when Service Launcher starts** option.
6. To make sure users must enter their password when logging in to WebCC, check the **Authentication** option.
7. By default, WebCC will use port **8080**. If you want to use a specific port to connect to WebCC, type it in the **Port** field.
8. Click on **Launch** to verify that WebCC launches successfully through Service Launcher using the selected parameters.
9. In the top menu, select **Server > Save All Settings**.

From now on, the WebCC server will launch through Service Launcher when you log in.



NOTE

You can enter parameters in the Other Parameters field to customize the WebCC server. For more information, see [Configuring WebCC with Custom Parameters on GNU/Linux on page 252](#)

Configuring WebCC with Custom Parameters on GNU/Linux

WebCC can run with no parameters, but it also provides some customization options. Among other things, you can change the port number, adjust its security settings and configure it to run with SSL security.

If you installed WebCC as a daemon, you can customize its parameters by modifying the shell script that launches WebCC, which the daemon launches. If you use Service Launcher to run WebCC, you can easily add parameters in the Other Parameters field of the WebCC tab of Service Launcher.

How to edit the WebCC shell script and add parameters

1. In a terminal, type in the following command:

```
$ sudo vi /usr/local/ToonBoomAnimation/harmony[Edition]_17/lnx86_64/bin/webcc.sh
```

2. Scroll all the way down to the following line:

```
./bin_3rdParty/node ../../cloud/server/app.js "$@"
```

3. Right after **app.js**, add a space, then type each parameter you want to add, separated by spaces. For example, if you want WebCC to use the port number **8081**, you would edit the line like so:

```
./bin_3rdParty/node ../../cloud/server/app.js -port 8081 "$@"
```




NOTE

For a list of parameters and their usage, see [WebCC Parameters Table on page 253](#).

4. Save and quit.
5. For your changes to take effect, restart the service with the following command:

```
$ sudo service toonboom_webcc restart
```

How to add parameters for WebCC in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the system tray, click on the Service Launcher  icon.

The Service Launcher application opens.

3. Select the **WebCC** tab.
4. If you want to let any user connect to WebCC without credentials and have administrative privileges, uncheck the **Authentication** option. Otherwise, make sure it is checked.
5. If you want to change the port used by the WebCC server, type the desired port number in the **Port** field.
6. If you want to specify any other option, type them in the **Other Parameters** field, separated by spaces.

**NOTE**


For a list of parameters and their usage, see [WebCC Parameters Table on page 253](#).

7. In the top menu, select **Server > Save All Settings**.
8. Click on the **Stop** button.
9. Click on the **Launch** button.

WebCC Parameters Table

The following is a table of parameters that can be specified to customize the behaviour of your WebCC server. They can be used in both the WebCC shell script and in Service Launcher.

Parameter	Description
<code>-numWorkers <number></code>	<p>The amount of server nodes to launch to process client requests. Each node requires one CPU core, and a single node can only process one client request at a time. Therefore, the more nodes are launched, the less likely the server is to stall under heavy load.</p> <p>If this parameter is not specified, WebCC will use as many nodes as there are CPU cores on the server.</p>
<code>-noAuth</code>	<p>Disables authentication. With this option enabled, any user accessing the WebCC web interface will be logged in as usabatch and will have administrative privileges on the entire database.</p>
<code>-port <port number></code>	<p>The port to which WebCC is deployed.</p> <p>The default port for accessing WebCC is 8080. For browsers to access WebCC without specifying the port number, you can use port 80.</p>

Parameter	Description
	<div style="border: 1px solid green; padding: 10px;">  <p>NOTE</p> <p>If either the <code>-http</code> or <code>-https</code> parameter is specified, the <code>-port</code> parameter will be ignored.</p> </div>
<code>-exchange <directory></code>	<p>The location of the exchange directory on the local filesystem. This directory is where all packages are stored when exporting or importing scenes.</p> <p>By default, this directory is <code>/USA_DB/exchange</code>.</p>
<code>-cacheDuration <seconds></code>	<p>The amount of seconds to keep data in the cache. This can be set to any value between 1 and 999999. By default, this is set to 3600.</p>
<code>-allowAllOrigin</code>	<p>Enables cross-origin resource sharing on the WebCC server.</p> <p>By default, as a standard security measure, only files on the same server as the WebCC server are allowed to use WebCC resources. With this option enabled, it is possible to create web interfaces on other servers that use resources stored on the WebCC server, or to store different modules of WebCC on different servers and let them communicate together.</p> <p>In most use cases, this option should be disabled to prevent other machines from exploiting resources on the WebCC server.</p>
<code>-disableAnalytics</code>	<p>Disables Google Analytics tracking for all clients. By default, this is enabled.</p>
<code>-permission <user type> <action> <on off></code>	<p>Overrides a default permission for the given user type.</p> <p>At the moment, this parameter can only be used for the user type <code>animator</code> and for the permission <code>import_export_any_scene</code>. Hence, when used with the following syntax, animator-type users can export and import any scene in the database, without requiring the scene to be assigned to them first.</p> <pre style="text-align: center;">-permission animator import_export_any_scene on</pre>

HTTPS/SSL Parameters Table


The following parameters can be used to enable and configure SSL for your WebCC server.



NOTE

For more information on how to configure WebCC with SSL, see [About SSL, HTTPS and WebCC on GNU/Linux on page 257](#).

Parameter	Description
<p><code>-https <port number></code></p>	<p>Enables connection to the server through the https protocol using the specified port number.</p> <p>A commonly used port for accessing WebCC through the https protocol is 8443. For web browsers to access WebCC via https without specifying the port number—provided that the address of the server is preceded by https://—you can use port 443.</p> <p>For https to work, you must either specify a pair of PEM format key and certificate with the -key and -cert option, or a PFX format certificate with the -pfx option.</p>
<p><code>-http <port number></code></p>	<p>If access through the https protocol is enabled, this parameter allows you to also allow access through the http protocol through the specified port.</p> <p>Using this parameter is optional. If the -https parameter is specified, but not the -http parameter, WebCC will only accept connections through https.</p> <p>The default port for accessing WebCC through the http protocol is 8080. For browsers to access WebCC via http without specifying the port number, you can use port 80.</p>
<p><code>-key <key file></code></p>	<p>The location of the private key file for the SSL session. This must be specified along with the -cert parameter.</p> <p>The private key must be in Privacy Enhanced Mail (PEM) format, which is an ASCII format where the key is encoded in Base64 and located between -----BEGIN PRIVATE KEY----- and -----END PRIVATE KEY----- tags. Typically private keys in this format have a .key extension.</p>
<p><code>-cert <certificate file></code></p>	<p>The location of the security certificate file for the SSL session. This must be specified along with the -key parameter.</p> <p>The certificate must be in Privacy Enhanced Mail (PEM) format, which is an ASCII format where the certificate is encoded in Base64 and located between -----BEGIN CERTIFICATE----- and -----END CERTIFICATE----- tags. Typically, such certificates have a .crt, .cer or .pem extension.</p>
<p><code>-pfx <pfx certificate file></code></p>	<p>If you have a Personal Information Exchange (PFX) certificate, you can use this parameter to specify its location, instead of using a PEM certificate and key. Typically, such certificates have a .pfx or .p12 extension. Since these certificates include their own private key, specifying a key with this certificate type is not required.</p>
<p><code>-ca <root ca certificate file></code></p>	<p>If you have a domain certificate that is signed by a certificate authority (CA), this parameter allows you to specify the root CA certificate and/or its intermediate</p>

Parameter	Description
	<p>certificates.</p> <p>If you want to specify several certificates so as to compose the chain going from your domain certificate to the root CA certificate, you can repeat the parameter, like so:</p> <pre data-bbox="643 464 1341 516">-ca <intermediate certificate 1> -ca <intermediate certificate 2> -ca <root CA certificate></pre> <p>Typically, you will be able to log on to WebCC using SSL from a Web browser even if WebCC is only configured with your domain certificate, provided that it is signed by a reputable certificate authority. However, if you want project collaborators to be able to use the Download Database Changes and Update Database Scene functionalities in Harmony via SSL, it may be necessary to specify the root CA certificates and/or its intermediates, as Harmony may not otherwise be able to recognize the authority of the CA who signed your certificate.</p> <p>Typically, you can obtain the root CA certificates or intermediate certificates from the root certificate authority of the certificate authority who signed your domain certificate.</p> <div data-bbox="565 940 1430 1150" style="border: 1px solid green; padding: 10px; margin-top: 10px;">  <p>NOTE</p> <p>If necessary, you can allow Harmony to trust the WebCC server even if it has a self-signed certificate—see Disabling SSL Certificate Validation for WebCC in Harmony on page 280.</p> </div>

About SSL, HTTPS and WebCC on GNU/Linux

You can configure your WebCC server to accept connections through the https protocol, either exclusively or alongside the http protocol. Connections through https are encrypted and the identity of the server is verified using a certificate. This ensures the information exchanged between the client and the server cannot be deciphered if intercepted. Also, provided that you configure WebCC with a certificate signed by a reputable certificate authority, this ensures that hackers cannot perform man-in-the-middle attacks by making their own server pose as your WebCC server to intercept its traffic.

For WebCC to accept connections with the https protocol, you must configure WebCC with the following:

- The port used for https connections.
- The private certificate key.
- The certificate.
- Optionally, the root CA and/or intermediate certificate.

There are two types of certificates supported by WebCC:

- **Privacy Enhanced Mail (PEM) certificates:** These certificates are ASCII files encoded in Base64. Typically, the certificate is in a file with a `.crt`, `.cer` or `.pem` extension and the private key is in a file with a `.key` extension.
- **Personal Information Exchange (PFX /PKCS #12) certificates:** These certificates are binary files that include both the certificate and its private key. Typically, such certificates are used for Windows servers and have a `.pfx` or `.p12` extension.

If your certificate is in any other format, such as DER, PKCS #7 or CA_bundle, you will need to perform extra steps to convert the certificate to a supported format.

About Certificate Signature

It is recommended to use a certificate that has been signed by a reputable certificate authority.

If your certificate is signed by a certificate authority, WebCC will typically be accessible without issue in a web browser. However, if you want clients to be able to upload and download scene changes directly between Harmony and the WebCC server, you may have to specify the root and/or intermediate certificate for your certificate. This is because web browsers are typically designed to trace a certificate's chain of trust all the way to the root certificate authority, whereas Harmony is not.

If your certificate is self-signed, web browsers will warn the user that the website is not secure, and users will need to go around that warning to access WebCC. Likewise, Harmony will not be able to upload or download scene changes directly from its interface at all, unless the client makes Harmony trust self-signed certificates by enabling the hidden preference `WEBCC_SSL_SELF_SIGNED`—see [Disabling SSL Certificate Validation for WebCC in Harmony on page 280](#).

Configuring WebCC for SSL Support on GNU/Linux

Like other WebCC options, configuring WebCC to use SSL is done by using command line parameters when launching WebCC. To use WebCC with SSL, you must specify the `-https` parameter followed by the port WebCC should use for https connections, the certificate and key file.

The following are examples of parameters to use for configuring WebCC to use SSL:

- If you have a PEM certificate and key file:

```
-https 8443 -key <path to PEM key file> -cert <path to PEM certificate>
```

- If you have a PEM certificate and its private key in the same file, specify the same file using both the `-key` and `-cert` parameters:

```
-https 8443 -key <path to PEM certificate> -cert <path to PEM certificate>
```

- If you have a PFX certificate, simply specify its path with the `-pfx` parameter:

```
-https 8443 -pfx <path to PFX certificate>
```

- If you want to allow users to connect to your server via either http or https, you can do so by specifying a port number for each protocol:

```
-http 8080 -https 8443 -key <path to key file> -cert <path to certificate>
```

- If you have a root certificate, an intermediate certificate, or both, you can specify each file preceded by the `-ca` parameter, in any order:

```
-http 8443 -key <path to key file> -cert <path to certificate> -ca <path to  
intermediate certificate 1> -ca <path to intermediate certificate 2> -ca <path to  
root certificate>
```

How to edit the WebCC shell script and add parameters

1. In a terminal, type in the following command:

```
$ sudo vi /usr/local/ToonBoomAnimation/harmony[Edition]_17/lnx86_  
64/bin/webcc.sh
```

2. Scroll all the way down to the following line:

```
./bin_3rdParty/node ../../cloud/server/app.js "$@"
```

3. Right after `app.js`, add a space, then type each parameter you want to add, separated by spaces. For example, if you want WebCC to use the port number **8081**, you would edit the line like so:

```
./bin_3rdParty/node ../../cloud/server/app.js -port 8081 "$@"
```


**NOTE**

For a list of parameters and their usage, see [HTTPS/SSL Parameters Table](#) on page 259.

4. Save and quit.
5. For your changes to take effect, restart the service with the following command:

```
$ sudo service toonboom_webcc restart
```

How to add parameters for WebCC in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the system tray, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **WebCC** tab.
4. If you want to let any user connect to WebCC without credentials and have administrative privileges, uncheck the **Authentication** option. Otherwise, make sure it is checked.
5. If you want to change the port used by the WebCC server, type the desired port number in the **Port** field.
6. If you want to specify any other option, type them in the **Other Parameters** field, separated by spaces.

**NOTE**


For a list of parameters and their usage, see [HTTPS/SSL Parameters Table](#) on page 259.

7. In the top menu, select **Server > Save All Settings**.
8. Click on the **Stop** button.
9. Click on the **Launch** button.

HTTPS/SSL Parameters Table

The following parameters can be used to enable and configure SSL for your WebCC server.

Parameter	Description
<code>-https <port number></code>	<p>Enables connection to the server through the https protocol using the specified port number.</p> <p>A commonly used port for accessing WebCC through the https protocol is 8443. For web browsers to access WebCC via https without specifying the port number—provided that the address of the server is preceded by https://—you can use port 443.</p> <p>For https to work, you must either specify a pair of PEM format key and certificate with the -key and -cert option, or a PFX format certificate with the -pfx option.</p>
<code>-http <port number></code>	<p>If access through the https protocol is enabled, this parameter allows you to also allow access through the http protocol through the specified port.</p> <p>Using this parameter is optional. If the -https parameter is specified, but not the -http parameter, WebCC will only accept connections through https.</p> <p>The default port for accessing WebCC through the http protocol is 8080. For browsers to access WebCC via http without specifying the port number, you can use port 80.</p>
<code>-key <key file></code>	<p>The location of the private key file for the SSL session. This must be specified along with the -cert parameter.</p> <p>The private key must be in Privacy Enhanced Mail (PEM) format, which is an ASCII format where the key is encoded in Base64 and located between -----BEGIN PRIVATE KEY----- and -----END PRIVATE KEY----- tags. Typically private keys in this format have a .key extension.</p>
<code>-cert <certificate file></code>	<p>The location of the security certificate file for the SSL session. This must be specified along with the -key parameter.</p> <p>The certificate must be in Privacy Enhanced Mail (PEM) format, which is an ASCII format where the certificate is encoded in Base64 and located between -----BEGIN CERTIFICATE----- and -----END CERTIFICATE----- tags. Typically, such certificates have a .crt, .cer or .pem extension.</p>
<code>-pfx <pfx certificate file></code>	<p>If you have a Personal Information Exchange (PFX) certificate, you can use this parameter to specify its location, instead of using a PEM certificate and key. Typically, such certificates have a .pfx or .p12 extension. Since these certificates include their own private key, specifying a key with this certificate type is not required.</p>
<code>-ca <root ca certificate file></code>	<p>If you have a domain certificate that is signed by a certificate authority (CA), this parameter allows you to specify the root CA certificate and/or its intermediate</p>

Parameter	Description
	<p>certificates.</p> <p>If you want to specify several certificates so as to compose the chain going from your domain certificate to the root CA certificate, you can repeat the parameter, like so:</p> <pre data-bbox="641 464 1339 516">-ca <intermediate certificate 1> -ca <intermediate certificate 2> -ca <root CA certificate></pre> <p>Typically, you will be able to log on to WebCC using SSL from a Web browser even if WebCC is only configured with your domain certificate, provided that it is signed by a reputable certificate authority. However, if you want project collaborators to be able to use the Download Database Changes and Update Database Scene functionalities in Harmony via SSL, it may be necessary to specify the root CA certificates and/or its intermediates, as Harmony may not otherwise be able to recognize the authority of the CA who signed your certificate.</p> <p>Typically, you can obtain the root CA certificates or intermediate certificates from the root certificate authority of the certificate authority who signed your domain certificate.</p> <div data-bbox="565 940 1429 1150" style="border: 1px solid #c8e6c9; padding: 10px; margin-top: 10px;"> <p> NOTE</p> <p>If necessary, you can allow Harmony to trust the WebCC server even if it has a self-signed certificate—see Disabling SSL Certificate Validation for WebCC in Harmony on page 280.</p> </div>

Setting Up Static IPs for WebCC on GNU/Linux

It is common to configure server machines to have a static IP. This avoids having to rely on DNS resolution to find the server when its IP address changes, and it also allows you to associate the hostname to the server's static IP address on the machine to skip DNS resolution altogether, increasing network performance.

If your machine is currently connected to a DHCP server that dynamically assigns IP addresses, you can configure your machine to permanently use its current IP address. To do this, you can start by making note of the information for your current network connection, then using these parameters for a permanent IP address.

How to obtain your current network configuration on CentOS 6

1. To obtain your machine's hostname, enter the following command:

```
$ hostname
```

Write down the output value.

2. To obtain the network domain, enter the following command:

```
$ hostname -d
```

Write down the output value.

3. To obtain your machine's default gateway and network device, enter the following command:

```
$ route
```

The output will look like this:

Destination	Gateway	Genmask	Flags	Metric	Ref	Use
Iface						
10.120.0.0	*	<subnet mask>	U	1	0	0
<device name>						
default	<default gateway>	0.0.0.0	UG	0	0	0
<device name>						

One of the rows will have **default** in the destination column. In this row, the name of the device in the **Iface** column is the name of your default network device. Write down the name of that device.

4. To obtain your machine's IP and subnet mask, enter the following command, replacing **<device name>** with the name of the device noted in the previous step:

```
$ nmcli -f all dev list iface <device name>
```

Information about the network interface will appear, in either one of the following formats:

GENERAL.DEVICE:	<device name>
GENERAL.TYPE:	802-3-ethernet
GENERAL.DRIVER:	tg3
GENERAL.HWADDR:	<MAC address>

```

GENERAL.STATE:                connected
CAPABILITIES.CARRIER-DETECT: yes
CAPABILITIES.SPEED:           1000 Mb/s
WIRED-PROPERTIES.CARRIER:    on
IP4-SETTINGS.ADDRESS:         <IPv4 address>
IP4-SETTINGS.PREFIX:          <prefix> (<subnet mask>)
IP4-SETTINGS.GATEWAY:         <default gateway>
IP4-DNS1.DNS:                  <DNS server address>

```

Write down the IPv4 address, the subnet mask, the default gateway as well as every DNS server address listed.



NOTE

You can also obtain the IP address (**inet addr**) and network mask (**Mask**) of your device with the command:

```
$ ifconfig <device name>
```

How to obtain your current network configuration on CentOS 7

1. To obtain your machine's hostname, enter the following command:

```
$ hostname
```

Write down the output value.

2. To obtain the network domain, enter the following command:

```
$ hostname -d
```

Write down the output value.

3. To obtain your machine's default gateway and network device, enter the following command:

```
$ route
```

The output will look like this:

Destination	Gateway	Genmask	Flags	Metric	Ref	Use
Iface						
10.120.0.0	*	<subnet mask>	U	1	0	0
<device name>						
default	<default gateway>	0.0.0.0	UG	0	0	0
<device name>						

One of the rows will have **default** in the destination column. In this row, the name of the device in the **Iface** column is the name of your default network device. Write down the name of that device.

4. To obtain your machine's IP and subnet mask, enter the following command, replacing **<device**

name> with the name of the device noted in the previous step:

```
$ nmcli -p -f IP4 connection show <device name>
```

Information about the network interface will appear. The output will be in this format.

```
=====
Activate connection details (0568a4c4-092a-46e9-9f25-2295a1389d6e)
=====
IP4.ADDRESS[1]:                <IPv4 address>/<prefix>
IP4.GATEWAY:                   <default gateway>
IP4.DNS[1]:                   <DNS server address>
IP4.DOMAIN[1]:                <search domains>
```

Write down the IPv4 address, the default gateway address, the DNS server address(es) and the search domain(s).

To obtain your subnet mask, enter the following command, replacing **<device name>** with the name of the device noted in the previous step:

```
$ nmcli -p -f DHCP4 connection show <device name> | grep 'subnet_mask'
```

The output should look like this:

```
DHCP4.OPTION[3]:                subnet_mask = <subnet mask>
```

Write down the subnet mask.



NOTE

You can also obtain the IP address (**inet**) and network mask (**netmask**) of your device with the command:

```
$ ifconfig <device name>
```

How to set up a static IP for CentOS 6 and 7

1. Before you do this, make sure you have the following information for your machine:
 - Name of default network device (ie: **eth0**, **p3p1**)
 - Hostname
 - Domain name
 - Search domain(s)
 - IPv4 address

- Network mask
 - Default gateway address
 - DNS server address(es)
2. Open your machine's network configuration file for editing:

```
$ sudo vi /etc/sysconfig/network
```

3. Configure the file as follows, replacing the bolded text with the appropriate values:

```
NETWORKING=yes  
HOSTNAME=<hostname>.<domain name>
```

4. Open your default network interface's configuration file for editing. This file will be named **ifcfg-** followed by the name of the network interface you noted earlier. For example, if your network interface is named **eth0**, the configuration file will be named **ifcfg-eth0**:

```
$ sudo vi /etc/sysconfig/network-scripts/ifcfg-<device name>
```

5. Configure the file with the appropriate values for the following parameters. If a value is already declared for one of these parameters, replace it. If not, add it to the file:

```
DEVICE="<device name>"  
BOOTPROTO=static  
IPADDR=<IPv4 address>  
NETMASK=<network mask>  
GATEWAY=<default gateway>  
ONBOOT=yes
```

6. Open the DNS configuration file for editing:

```
$ sudo vi /etc/resolv.conf
```

7. Configure the file as follows, replacing the bolded text with the appropriate values:

```
search <search domain names>  
nameserver <DNS server address>
```



NOTES

- If you have several search domains, enter them all on the same line after **search**, separated by spaces.
- If you have several DNS servers, add one **nameserver** line for each address.

8. Restart the network service:

```
$ sudo service network restart
```

Opening the Firewall Ports for a WebCC Server on GNU/Linux

The GNU/Linux firewall will block attempts from other computers to communicate with the WebCC server. In order to circumvent this, the ports used by the database server must be opened.

How to allow connections to the WebCC server to go through the firewall on CentOS 6, RedHat Enterprise Linux 6 or other systems using the iptables service

1. Add the ports used by the WebCC server to the list of open ports with the following commands:

```
$ sudo iptables -I INPUT -m state --state NEW -p tcp -m tcp --dport 8080 -j ACCEPT
$ sudo service iptables save
$ sudo service iptables restart
```

The GNU/Linux firewall should now allow other machines to connect to WebCC.



NOTES

- 8080 is the default port used by the WebCC server. You can customize this port number—see [Configuring WebCC with Custom Parameters on GNU/Linux on page 252](#).
- If your WebCC server is configured to allow both http and https connections, you will need to add a firewall rule for the port used by both protocols.

How to allow connections to the the WebCC server to go through the firewall on CentOS 7, RedHat Enterprise Linux 7 or other systems using the firewalld service

1. Add the ports used by the WebCC server to the list of open ports with the following commands:

```
$ sudo firewall-cmd --permanent --zone=public --add-port=8080/tcp
$ sudo firewall-cmd --reload
```

The GNU/Linux firewall should now allow other machines to connect to WebCC.



NOTES

- 8080 is the default port used by the WebCC server. You can customize this port number—see [Configuring WebCC with Custom Parameters on GNU/Linux on page 252](#).
- If your WebCC server is configured to allow both http and https connections, you will need to add a firewall rule for the port used by both protocols.

Checking the WebCC Server Log on GNU/Linux


In order to troubleshoot problems with a WebCC server, you can look through its log. This is done differently depending on whether you installed the database server as a daemon or using Service Launcher.

How to check the log for the WebCC server daemon

1. In a terminal, enter the following command:

```
$ less /tmp/webccserver.log
```

How to check the log for the WebCC server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the system tray, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **WebCC** tab.
4. In the bottom half, select the **Log** tab.
5. Click on the **View Complete Log** button in the bottom-right corner.

Launching the WebCC Server Manually on GNU/Linux

You can start the WebCC server directly from a terminal. This is usually not practical for a production environment, but it can be very useful when troubleshooting problems. You can view the output of the server directly in the command line as it launches and handles requests.

How to start the WebCC server manually

1. In a command line, enter the following command:

```
$ /usr/local/ToonBoomAnimation/harmony[Edition]_17/lnx86_64/bin/webcc.sh
```

Stopping and Restarting the WebCC Server on GNU/Linux

If you installed the WebCC server as a daemon, you can stop it and restart it using the `service` command line tool. If you installed it using Service Launcher, you can stop it and restart it manually in Service Launcher.

How to stop and restart the WebCC server daemon

1. In a terminal, enter the following command:

```
$ sudo service toonboom_webcc stop
```

2. When you are ready to start the server again, enter the following command:

```
$ sudo service toonboom_webcc start
```




TIP

You can also restart the service instantly with the following command:

```
$ sudo service toonboom_webcc restart
```

How to stop and restart the WebCC server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the system tray, click on the Service Launcher  icon.

The Service Launcher application opens.

3. Select the **WebCC** tab.
4. Click on **Stop**.
5. When you are ready to start the server again, click on **Launch**.

Uninstalling the WebCC Server on GNU/Linux

If you installed the WebCC server as a daemon, you can uninstall it using the `chkconfig` command line tool. If you installed it using Service Launcher, you can uninstall it in Service Launcher.


How to disable the WebCCserver daemon

1. In a terminal, type in the following commands to stop, then disable the daemon:

```
$ sudo service toonboom_webcc stop
$ sudo chkconfig toonboom_webcc off
```

The WebCC server daemon will stop and will no longer launch on boot.

How to uninstall the WebCC server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the system tray, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **WebCC** tab.
4. Click on **Stop**.
5. Uncheck the **Launch when Service Launcher starts** option.
6. In the top menu, select **Server > Save All Settings**.

Configuring Render Nodes for WebCC Servers

In order to be able to generate small preview movies that can be played in a web browser for the scenes on your Harmony database, the render nodes for your Harmony database must have the Cisco OpenH264 library installed.

Because of copyright restrictions, this library is not installed automatically with Harmony. It must be downloaded off the Cisco server as it is installed. However, regardless of the platform on which you are installing WebCC, a configuration tool is provide to automate the procedure.



IMPORTANT

- The Cisco OpenH264 library must be installed on each machine on your database server's local network that will be configured to render scenes for your database.
- Installing the Cisco OpenH264 library requires an Internet connection.
- Further steps are required in order for a machine to be configured to render preview movies. See [Batch Processing](#).

Installing the Cisco OpenH264 Codec for WebCC on Windows	273
Installing the Cisco OpenH264 Codec for WebCC on macOS	274
Installing the Cisco OpenH264 Codec for WebCC on GNU/Linux	275

Installing the Cisco OpenH264 Codec for WebCC on Windows


Batch processing machines must have the Cisco OpenH264 library installed in order to be able to generate small preview movies for WebCC. Because of copyright restriction, this codec must be downloaded off Cisco's servers as it is installed. The Configuration Wizard can automatically download it and copy it in Harmony's installation directory.



NOTE

Installing the Cisco OpenH264 library requires a working Internet connection.

How to install the Cisco OpenH264 codec for WebCC

1. In the **Start** menu, open the **Programs** menu and select **Harmony 17 [Edition] > Tools >  Configuration Wizard**.
2. When prompted, enable Configuration Wizard to make changes to your device.
The Toon Boom Harmony Configuration Wizard launches.
3. In the list of tasks, right of **Cisco OpenH264 Library**, click on
4. In the ensuing dialog, under **Do you wish to install the OpenH264 Library?**, select **Yes/Install**.
5. Click on **Next**.
6. Click on **Finish** to confirm the changes.

The machine should now be able to generate preview movies for WebCC.



IMPORTANT

In order to be able to generate preview movies for WebCC, a machine must also be configured to be a batch processing node for your database. For more information, see [Batch Processing](#).

Installing the Cisco OpenH264 Codec for WebCC on macOS


Batch processing machines must have the Cisco OpenH264 library installed in order to be able to generate small preview movies for WebCC. Because of copyright restriction, this codec must be downloaded off Cisco's servers as it is installed. The Configuration Assistant can automatically download it and copy it in Harmony's installation directory.



NOTE

Installing the Cisco OpenH264 library requires a working Internet connection.

How to install the Cisco OpenH264 codec for WebCC

1. In Finder, proceed to **Applications >  Toon Boom Harmony 17 [Edition]** and double-click **Harmony [Edition]**.
The main Harmony application launches.
2. In the top menu, select **Harmony [Edition] > Tools > Configuration Assistant**.
3. Check the **Install/Uninstall Cisco OpenH264 Library** option and uncheck all other options.
4. Under **Choose to install or uninstall**, select **Install OpenH264 Library**.
5. Click on **Create**.
6. When prompted, enter the username and the password of a user with administrator privileges.

The machine should now be able to generate preview movies for WebCC.



IMPORTANT

In order to be able to generate preview movies for WebCC, a machine must also be configured to be a batch processing node for your database. For more information, see [Batch Processing](#).

Installing the Cisco OpenH264 Codec for WebCC on GNU/Linux

Batch processing machines must have the Cisco OpenH264 library installed in order to be able to generate small preview movies for WebCC. Because of copyright restriction, this codec must be downloaded off Cisco's servers as it is installed.

There are two ways to installing the Cisco OpenH264 library, both of which will automatically download the library and store it in the Harmony installation directory:

- By running the `openh264_control.sh` script packaged with Harmony
- By running the Harmony installation script with the `-openh264` parameter.



NOTE

Installing the Cisco OpenH264 library requires a working Internet connection.

How to install the Cisco OpenH264 codec for WebCC using the script packaged with Harmony

1. In a terminal, type in the following commands:

```
$ cd /usr/local/ToonBoomAnimation/harmony[Edition]_17/lnx86_64/bin
$ sudo ./openh264_control.sh
```

2. When prompted, type `i` and press enter.

The machine should now be able to generate preview movies for WebCC.



IMPORTANT

In order to be able to generate preview movies for WebCC, a machine must also be configured to be a batch processing node for your database. For more information, see [Batch Processing](#).

How to enable the WebCC server daemon using the Harmony installation script

1. In a terminal, navigate to the directory where you downloaded the installation package.

```
$ cd ~/Downloads
```

2. Extract the package's contents in the directory.

```
$ tar -xvzf ./name-of-package.tar.gz
```

3. Switch to the directory in which the package's contents were extracted.

```
$ cd name-of-package
```

4. Run the installer with the `-d` parameter:

```
$ sudo ./install -openh264
```

5. When prompted, confirm that you want to proceed with the installation.

The machine should now be able to generate preview movies for WebCC.

**IMPORTANT**

In order to be able to generate preview movies for WebCC, a machine must also be configured to be a batch processing node for your database. For more information, see [Batch Processing](#).

Configuring Local WebCC Clients

Since WebCC is a web-based platform, the great majority of its functionalities are available to any computer that has access to the WebCC server through the web and has a compatible web browser.

However, while users who access the WebCC server from a remote location must download scenes so that they can work on them locally, users who access the WebCC server from the same local network as the Harmony database server can open scenes directly from WebCC into Harmony, without download them first.

To be able to this, the client machine must have Harmony Server installed and must be configured as a Harmony database client—see [Connecting Client Workstations to a Harmony Database Server on page 150](#).

On **GNU/Linux** the registration of the WebCC URLs to open scenes in Harmony is not automatic. It must be specifically enabled when installing Harmony.

Configuring a GNU/Linux Workstation to Open Harmony Scenes from WebCC278

Configuring a GNU/Linux Workstation to Open Harmony Scenes from WebCC

The GNU/Linux installer can configure your machine to open **harmony://** URLs to open in Harmony. Such URLs are used by WebCC to open scenes directly in Harmony.

How to register the **harmony://** URL handler on a GNU/Linux workstation

1. In a terminal, navigate to the directory where you downloaded the installation package.

```
$ cd ~/Downloads
```

2. Extract the package's contents in the directory.

```
$ tar -xvzf ./name-of-package.tar.gz
```

3. Switch to the directory in which the package's contents were extracted.

```
$ cd name-of-package
```

4. Launch the installer with the **-s** parameter:

```
$ sudo ./install -s
```

5. When prompted, confirm that you want to proceed with the installation.

The **harmony://** URL handler is now registered to launch Harmony and open Harmony scenes.



NOTE

You may need to reboot the machine before this change takes effect.

Configuring Remote WebCC Clients

Since most of the functionalities of WebCC are web-based, there is very little configuration to do on client machines. A machine that has access to the WebCC server and has a compatible web browser can use all the web-based features of WebCC, can download scene packages.

However, when working on a scene exported from WebCC in Harmony, it is possible to download changes made to the scene directly into the scene via Harmony. Likewise, it is also possible to upload changes that you made to the scene directly to the WebCC server from Harmony.

In order to do this, you must configure Harmony to communicate with the WebCC server. This is as simple as typing in the full address of the server, as you do in your web browser.



NOTE

You will be prompted for your WebCC username and password when attempting to download changes from WebCC or to update a scene to WebCC.

How to configure Harmony to communicate with WebCC

1. In the top menu, select **Edit (Windows/Linux) or Harmony [Edition] (macOS) > Preferences**.
2. Open the **General** tab.

In the bottom-right corner of the tab, there should be a **Web Control Center** section.



3. In the **URL** field of the Web Control Center section, type in the URL of the WebCC server including the protocol, domain and port number, as you would in your web browser:

`protocol://hostname:portnumber`

For example, to access a server named **webcc.mystudio** through **https** via port **8843**, you would enter:

`https://webccserver.mystudio:8843`

4. Click on the **OK** button.

Disabling SSL Certificate Validation for WebCC in Harmony

If your WebCC server uses as self-signed certificate to support SSL, Harmony will not be able to download scene changes from the WebCC server nor to upload scene changes to the WebCC server. To work around this, you can enable a hidden preferences that will allow Harmony to communicate with the WebCC server without validating its SSL certificate.



IMPORTANT

If this option is enabled, Harmony will not be able to verify the identity of your WebCC server before it communicates with it, which may cause security vulnerabilities. This should only be done for debugging purposes or if you are set on using a self-signed certificate for your WebCC server.

How to disable SSL Certificate Validation in Harmony

1. Open the Script Editor view by doing one of the following:
 - In the top-right corner of an existing view (ie: the Camera or Drawing view), click on the Add View **+** button and select **Script Editor**.
 - In the top menu, select **Windows > Script Editor**.
2. In the File list, select < < **Sandbox** > >.
3. In the script field, copy and paste the following code:

```
preferences.setBool("WEBCC_SSL_SELF_SIGNED", true);
```

4. In the Script Editor toolbar, click on Run Script

Chapter 4: Batch Processing

Harmony Server allows you to use batch processing to automatically delegate vectorization and rendering tasks to different computers, which will perform them in the background, so as to optimize your network's resources and avoid making project collaborators use their own workstation to render scenes, which can stall their machine by using up all its resources during the process.

In other words, batch processing allows you to create a render farm using the machines that are connected to your Harmony database server.

There are two types of tasks performed by batch processing:

Vectorizing

Vectorization converts scanned drawings to vector drawings. Scanned drawings are stored in bitmap format, and must be converted to a vector-based format before they can be painted in Harmony. In a traditional animation pipeline, this step is required before you can start painting scanned drawings.

When scanning a drawing using the Harmony [Edition] application, they are automatically sent to the vectorize queue of the database. It may sometimes be required to send drawings for vectorization manually in Control Center, for example in the following cases:

- To processing images that were not scanned in the Scan module.
- To recover the unpainted drawings after making an unrecoverable mistake while painting or post-processing the scene.
- To vectorize a pencil test.

You can open any specific job's vectorize queue in Control Center to check on the status of vectorization jobs.

Rendering

Also known as *compositing*, rendering is the process of creating final frames for a scene and storing them as image files as well as movie files. Rendered images have all their effects composited and are created in high quality, as they should be used for post-production and broadcasting.

Rendering a scene is a resource intensive process, and it is impractical for animators to use their own machines to render their scenes after working on them, as the rendering process is likely to stall their machine and prevent them from working on their scene any further until it is completed. Hence, by setting up batch processing, they can delegate the rendering process by sending their scenes to the rendering queue. Machines configured to perform batch processing will take care of performing the rendering job when they are ready and scheduled to do so.

You can open any specific job's render queue in Control Center to check on the status of rendering jobs.

Configuring Batch Processing

In order to configure batch processing for a Harmony database, machines that must perform batch processing tasks must be configured to do so. This configuration procedure can be broken down in three basic steps:

1. Add the name of the machine that will perform batch processing to the Harmony database's machine list.
2. Install the batch processing service on this machine. This service will run in the background and monitor the database for batch processing tasks to perform, then process these tasks in the background.
3. Create a batch processing schedule for this machine. The machine must be on a batch processing schedule in order to perform batch processing. Any single machine's schedule can be unrestricted, which means the machine always performs batch processing tasks if any are on queue, or can be restricted to specific hours, specific environments and specific tasks. It is recommended to schedule machines to perform renders outside of business hours so as to avoid impeding project collaborators by taking up their machine's resources while they are working.



IMPORTANT

Configuring batch processing requires using command line tools frequently. It is highly recommended to add the path to Harmony's command line tools to your machine's **PATH** environment variable so that you do not have to type the full path to these command line tools in the terminal.

To add Harmony to your path, follow the steps in the following topics:

- [Adding Harmony Executables to the %PATH% Environment Variable on Windows on page 18](#)
- [Adding Harmony Binaries to the \\$PATH Environment Variable on macOS on page 22](#)
- [Adding Harmony Binaries to the \\$PATH Environment Variable on GNU/Linux on page 38](#)

Creating machine-list Files

The `machine-list` file identifies all the machines on your network that can be allowed to batch process Harmony scenes. The machines on this list represents your network's render farm.



NOTE

You must have sufficient batch processing licenses to run batch processing on all the machines in the `machine-list` file.

How to create the machine-list file

1. In a text editor, create a new file at this location

```
/USA_DB/schedules/machine-list
```

2. In this text file, type in the hostname of every machine that you want to use for batch processing, one machine per line.

For example, you might have a machine-list, where each line is the name of a computer on your network:

```
anim-1
anim-2
anim-3
paint-1
paint-2
paint-3
layout-1
layout-2
```



NOTES

- There must be no blank lines in the machine-list file.
- The order in which machines are listed determines their order in the batch processing scheduling commands.
- You can obtain the name for a machine by typing `hostname` in a terminal or command prompt, regardless of the operating system.

3. Save the file.

The next time you use a scheduling command, it will be able to check up on or update the schedule of any machine in the machine list file.

Batch Processing Installation

The main steps involved in installing batch processing creating the **machine-list** file and to install the batch processing service.



IMPORTANT

Make sure the Cisco OpenH264 codec is installed on all the machines that will do the batch render of movie previews—see [Installing WebCC on Windows on page 196](#), [Installing WebCC on macOS on page 223](#) and [Installing WebCC on GNU/Linux on page 245](#).

Installing the Batch Processing Service on Windows

On Windows, you can install the batch processing service using the Configuration Wizard to make it launch on boot as user `usabatch`, or Service Launcher to make it launch when the current user logs in.

Installing the Batch Processing Service on Windows	287
Running the Batch Processing Service on Log-In on Windows	288
Looking Up the Batch Processing Log on Windows	290
Launching Batch Processing Manually on Windows	291
Stopping and Restarting the Batch Processing Service on Windows	292
Interrupting a Batch Processing Task on Windows	294
Uninstalling the Batch Processing Service on Windows	295

Installing the Batch Processing Service on Windows


You can easily install batch processing as a Windows service using the Configuration Wizard. It will launch as soon as the machine boots, under user **usabatch**.



NOTE

The following steps indicate how to make batch processing launch on boot, via the user **usabatch**, who will have administrator privileges. If you prefer, you can make the database server launch on log-in as the logged-in user. For more information, see [Running the Batch Processing Service on Log-In on Windows on page 288](#).

How to install the batch processing service

1. In the **Start** menu, open the **Programs** menu and select **Harmony 17 [Edition] > Tools >  Configuration Wizard**.
2. When prompted, enable Configuration Wizard to make changes to your device.
The Toon Boom Harmony Configuration Wizard launches.
3. In the list of tasks, right of **Batch Processing**, click on
4. In the ensuing dialog, under **Will this computer perform batch processing?**, select **Yes**.
5. Click on **Next**.
6. Under **What is the name of the server?**, type in the name of the database server.
7. Under **What is the User?**, type in the username
8. Click on **Finish** to confirm the changes.

The batch processing service is now installed and should be running. You can test that your machine is ready to perform batch processing job by entering the following command in a terminal, replacing **<host name>** with the name of your machine.

```
> Status -b -m <host name>
```


Running the Batch Processing Service on Log-In on Windows

The traditional way of running the batch processing service is by running the server as a Windows service or daemon. This means the service launches in the background as soon as the machine boots, and starts processing the queue even if nobody is logged in to the computer.

Using Service Launcher, it is possible to configure the batch processing service to launch as the currently logged-in user, when they log in. This means that your machine will only process jobs when the user is logged in.

Although this may seem impractical, it may be preferable for security reasons. Harmony services are run under the user `usabatch`, an user with administrator privileges. When using Service Launcher, the logged-in user does not need to have administrator privileges to run services.


Before configuring Service Launcher to run the batch processing service, the Windows service that launches it must be disabled, so as to avoid launching two instances of the batch processing service simultaneously.



NOTE

While daemons run regardless of who is logged in and even when no user is logged-in, Service Launcher will only run when the user for which it was specifically configured is logged in. This means that if you want Service Launcher to launch when any user is logged in, you must configure it for each user that has access to the machine.

How to disable the batch processing service on Windows

1. Open the **Start** menu and, in the **Programs** menu, select **Harmony 17 [Edition] > Tools >  Control Panel**.
2. When prompted, allow Control Panel to make changes to your device.
3. In the **Process Starter** drop-down, select **HarmonyStarter0-tbprocess -schedule**.



NOTE

The number after **HarmonyStarter** may be different, depending on how many Harmony services were installed before this one.

4. Click on **Stop**.
5. Click on **Remove**.

From that point on, the batch processing service will no longer launch on boot.

6. Close the **Control Panel**.



How to configure the batch processing service to launch on user log-in

1. Make sure you are currently signed-in as the user who will run the service.



NOTE

This user will need to be logged in for the service to be running. The user does not need to have administrator privileges.

2. Do one of the following:
 - If Service Launcher is not running, open the **Start** menu and, in the **Programs** list, select **Harmony 17 [Edition] > Tools >  Service Launcher**.
 - If Service Launcher is already running, click on its icon  in the notification tray.

The Service Launcher application launches.

3. In the **Application parameters** section of the **Preferences** tab, make sure that the **Launch on login** option is checked, so that services configured in Service Launcher automatically launch when you log in to your account.
4. Select the **ToonBoom Process** tab.
5. Make sure the **Schedule** option is checked.
6. Click on **Launch** to verify that the batch processing service launches successfully through Service Launcher.
7. Check the **Launch when Service Launcher starts** option.
8. In the top menu, select **Server > Save All Settings**.

From now on, the batch processing service will launch through Service Launcher when you log in. You can test that your machine is ready to perform batch processing job by entering the following command in a terminal, replacing **<host name>** with the name of your machine.

```
> Status -b -m <host name>
```


Looking Up the Batch Processing Log on Windows

In order to troubleshoot problems with the batch processing service, you can look through its log. This is done differently depending on whether you installed batch processing as a Windows service using Configuration Wizard or using Service Launcher.

How to check the log for the batch processing service

1. Open an Explorer window.
2. Browse to the following location:
C:\ProgramData\Toon Boom Animation\Toon Boom Harmony
3. Right-click on **tbprocess.log** and select **Open with > Notepad**.

How to check the log for the batch processing service in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the notification tray, click on the  Service Launcher icon.
The Service Launcher application opens.
3. Select the **ToonBoom Process** tab.
4. In the bottom half, select the **Log** tab.
5. Click on the **View Complete Log** button in the bottom-right corner.

Launching Batch Processing Manually on Windows

You can start the batch processing service directly from a command prompt. This is usually not practical for a production environment, but it can be very useful when troubleshooting problems. You can view the output of the service directly in the terminal in which you launch it as it starts and handles jobs.

How to start the batch processing service manually


1. In a command prompt, enter the following command:

```
$ C:\Program Files (x86)\Toon Boom Animation\Toon Boom Harmony 17  
[Edition]\win64\bin\tbprocess -schedule
```

Stopping and Restarting the Batch Processing Service on Windows

If you installed the batch processing service as a Windows service using Configuration Wizard, you can stop it and restart it using the Harmony Control Panel or the Windows Services control panel. If you installed it using Service Launcher, you can stop it and restart it manually in Service Launcher.

How to stop the batch processing service using the Harmony Control Panel

1. Open the **Start** menu and, in the **Programs** menu, select **Harmony 17 [Edition] > Tools >  Control Panel**.
2. When prompted, allow Control Panel to make changes to your device.
3. In the **Process Starter** drop-down, select **HarmonyStarter0-tbprocess -schedule**.



NOTE

The number after **HarmonyStarter** may be different, depending on how many Harmony services were installed before this one.

4. Click on **Stop**.
5. When you are ready to start the server again, click on **Start**.

How to stop the batch processing service using the Windows Services control panel

1. Open the Windows Control Panel:
 - On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
 - On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.
2. In the Control Panel, navigate to **System and Security > Administrative Tools**.
3. Double-click on **Services**.
4. Scroll-down to **HarmonyStarter0-tbprocess -schedule**.




NOTE

The number after **HarmonyStarter** may be different, depending on how many Harmony services were installed before this one.

5. Right-click on the service and select **Stop**.
6. When you are ready to start the server again, right-click on it and select **Start**.

How to stop and restart the batch processing service in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the notification tray, click on the  Service Launcher icon.
The Service Launcher application opens.
3. Select the **ToonBoom Process** tab.
4. Click on **Stop**.
5. When you are ready to start the server again, click on **Launch**.

Interrupting a Batch Processing Task on Windows

Control Center will allow you to cancel a batch processing task before it starts by deleting it from the queue, but if you delete a batch processing task while it is already processing, it will still finish in the background.

Vectorizing tasks are added to the queue on a per-drawing basis. This means that if you want to interrupt the vectorization of a scene, you can simply stop the batch processing service and clear the queue in Control Center. However, rendering tasks are added to the queue on a per-scene basis. Therefore, the only way to interrupt the rendering of a scene while it is currently rendering is by interrupting the Harmony process that is rendering the scene.



IMPORTANT

You should only interrupt a batch processing task when it is necessary, such as when an extremely lengthy task was started by accident and is liable to delay the other tasks.

How to interrupt a batch processing job

1. Press Ctrl + Alt + Del to open the Windows security screen.
2. Select Task Manager.
3. If the Task Manager only lists the open applications and has no menu or tabs, select **More details** at the bottom.
4. Open the **Details** tab.
5. Look for a process named **tbprocess.exe**.
6. Right-click on the process and select **End process tree**.

This will terminate the batch processing daemon (**tbprocess.exe**) as well as any Harmony process it might have started to render a scene or vectorize drawings. The batch processing daemon should restart instantly, but it will not attempt to process the batch processing task that was interrupted again.




NOTE

The job you interrupted will still appear in the queue with the **Processing** status in Control Center. You will need to delete the job manually from the queue to clear it up.


Uninstalling the Batch Processing Service on Windows

If you installed the batch processing service as a Windows service using Configuration Wizard, you can uninstall it using the Configuration Wizard, the Harmony Control Panel or the Windows Services control panel. If you installed it using Service Launcher, you can uninstall it in Service Launcher.

How to uninstall the batch processing service using Configuration Wizard

1. In the **Start** menu, open the **Programs** menu and select **Harmony 17 [Edition] > Tools >  Configuration Wizard**.
2. When prompted, enable Configuration Wizard to make changes to your device.
The Toon Boom Harmony Configuration Wizard launches.
3. In the list of tasks, right of **Batch Processing**, click on
4. In the ensuing dialog, under **Will this computer perform batch processing?**, select **No**.
5. Click on **Next**.
6. Click on **Finish** to confirm the changes.
The batch processing service will stop and will no longer launch on boot.

How to uninstall the batch processing service using the Harmony Control Panel

1. Open the **Start** menu and, in the **Programs** menu, select **Harmony 17 [Edition] > Tools >  Control Panel**.
2. When prompted, allow Control Panel to make changes to your device.
3. In the **Process Starter** drop-down, select **HarmonyStarter0-tbprocess -schedule**.



NOTE

The number after **HarmonyStarter** may be different, depending on how many Harmony services were installed before this one.

4. Click on **Stop**.
5. Click on **Remove**.
The batch processing service will stop and will no longer launch on boot.
6. Close the **Control Panel**.

How to uninstall the WebCC server service using the Windows Services control panel

1. Open the Windows Control Panel:
 - On **Windows 7**: Open the Start menu and select **All Programs > Accessories > System Tools > Control Panel**.
 - On **Windows 8.1** or **Windows 10**: Open the Start menu and, in the programs list, select **Windows System > Control Panel**.
2. In the Control Panel, navigate to **System and Security > Administrative Tools**.
3. Double-click on **Services**.
4. Scroll-down to **HarmonyStarter0-tbprocess -schedule**.




NOTE

The number after **HarmonyStarter** may be different, depending on how many Harmony services were installed before this one.

5. Double-click on the service to open its Properties dialog.
6. In the **Startup type** drop-down, select **Disabled**.
From that point on, the batch processing service will no longer launch on boot.
7. Click on **OK**.

How to uninstall the WebCC server in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the notification tray, click on the  Service Launcher icon.
The Service Launcher application opens.
3. Select the **ToonBoom Process** tab.
4. Click on **Stop**.
5. Uncheck the **Launch when Service Launcher starts** option.
6. In the top menu, select **Server > Save All Settings**.

Installing the Batch Processing Service on macOS

On macOS, you can install the batch processing service using the Configuration Assistant to make it launch on boot as user `usabatch`, or Service Launcher to make it launch when the current user logs in.

Installing the Batch Processing Daemon on macOS	298
Running the Batch Processing Service on Log-In on macOS	300
Looking Up the Batch Processing Log on macOS	302
Launching Batch Processing Manually on macOS	303
Stopping and Restarting the Batch Processing Daemon on macOS	304
Interrupting a Batch Processing Task on macOS	305
Uninstalling the Batch Processing Daemon on macOS	307

Installing the Batch Processing Daemon on macOS

On macOS, you can install the batch processing daemon using the Configuration Assistant.



NOTE

The following steps indicate how to make batch processing launch on boot, via the user `usabatch`, who will have administrator privileges. If you prefer, you can make the database server launch on log-in as the logged-in user. For more information, see [Running the Batch Processing Service on Log-In on macOS on page 300](#).

How to create the user `usabatch` on macOS

1. Open the Apple menu and select **System Preferences**.
2. Click on **Users and Groups**.
3. In the bottom-left corner of the Users & Groups dialog, click on the lock icon labeled **Click the lock to make changes** to get administrator permissions.
4. Enter the username and password of an account with administrator privileges.
5. Click on the + button at the bottom of the user list.
The user creation panel appears.
6. In the **New Account** drop-down, select **Administrator**.
7. In the **Full Name** and **Account Name** fields, enter `usabatch`.
8. Give the user a password.
9. Click on **Create User**.

How to install the database server daemon on macOS

1. In Finder, proceed to **Applications > H Toon Boom Harmony 17 [Edition]** and double-click **Harmony [Edition]**.
The main Harmony application launches.
2. In the top menu, select **Harmony [Edition] > Tools > Configuration Assistant**.
 1. In the Welcome screen, check the **Create the Startup Items** option and uncheck all other options.
 2. Click on Continue.
 3. Check the **Create Queue Server Startup Item** option and uncheck all other options.

4. Click on **Create**.
5. When prompted, enter the username and the password of a user with administrator privileges.

The batch processing daemon is now installed and should be running. You can test that your machine is ready to perform batch processing job by entering the following command in a terminal, replacing **<host name>** with the name of your machine.

```
$ Status -b -m <host name>
```

Running the Batch Processing Service on Log-In on macOS

The traditional way of running the batch processing service is by running the server as a Windows service or daemon. This means the service launches in the background as soon as the machine boots, and starts processing the queue even if nobody is logged in to the computer.

Using Service Launcher, it is possible to configure the batch processing service to launch as the currently logged-in user, when they log in. This means that your machine will only process jobs when the user is logged in.

Although this may seem impractical, it may be preferable for security reasons. Harmony services are run under the user `usabatch`, an user with administrator privileges. When using Service Launcher, the logged-in user does not need to have administrator privileges to run services.

Before configuring Service Launcher to run the batch processing service, the windows service that launches it must be disabled, so as to avoid launching two instances of the WebCC server simultaneously.



NOTE

While daemons run regardless of who is logged in and even when no user is logged-in, Service Launcher will only run when the user for which it was specifically configured is logged in. This means that if you want Service Launcher to launch when any user is logged in, you must configure it for each user that has access to the machine.

How to disable the batch processing daemon on macOS

1. In a terminal, enter the following command:

```
$ sudo launchctl unload  
/Library/LaunchDaemons/com.toonboom.QueueServer.plist
```

The batch processing daemon will stop and will no longer launch on boot.

How to configure the batch processing daemon to launch on user log-in


1. Make sure you are currently signed-in as the user who will run the service.




NOTE

This user will need to be logged in for the service to be running. The user does not need to have administrator privileges.

2. Do one of the following:

- If Service Launcher is not running, follow these steps to launch it:
 - a. In Finder, proceed to **Applications >  Toon Boom Harmony 17 [Edition]** and double-click **Harmony [Edition]**.

The main Harmony application launches.
 - b. In the top menu, select **Harmony [Edition] > Tools > Service Launcher**.
- If Service Launcher is already running, click on its icon  in the menu bar.

The Service Launcher application window appears.

3. In the **Application parameters** section of the **Preferences** tab, make sure that the **Launch on login** option is checked, so that services configured in Service Launcher automatically launch when you log in to your account.
4. Select the **ToonBoom Process** tab.
5. Make sure the **Schedule** option is checked.
6. Click on **Launch** to verify that the batch processing service launches successfully through Service Launcher.
7. Check the **Launch when Service Launcher starts** option.
8. In the top menu, select **Server > Save All Settings**.

From now on, the batch processing service will launch through Service Launcher when you log in. You can test that your machine is ready to perform batch processing job by entering the following command in a terminal, replacing **<host name>** with the name of your machine.

```
$ Status -b -m <host name>
```

Looking Up the Batch Processing Log on macOS


In order to troubleshoot problems with the batch processing service, you can look through its log. This is done differently depending on whether you installed the database server as a daemon using Configuration Assistant, or using Service Launcher.

How to check the log for the batch processing daemon

1. In a terminal, enter the following command:

```
$ less /tmp/com.toonboom.QueueServer.log
```

How to check the log for the batch processing daemon in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the top menu, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **ToonBoom Process** tab.
4. In the bottom half, select the **Log** tab.
5. Click on the **View Complete Log** button in the bottom-right corner.

Launching Batch Processing Manually on macOS

You can start the batch processing service directly from a terminal. This is usually not practical for a production environment, but it can be very useful when troubleshooting problems. You can view the output of the service directly in the terminal in which you launch it as it starts and handles jobs.

How to start the batch processing service manually

1. In a terminal, enter the following command:

```
$ "/Applications/Toon Boom Harmony 17 [Edition]/Harmony  
[Edition].app/Contents/tba/macosx/bin/tbprocess" -schedule
```


Stopping and Restarting the Batch Processing Daemon on macOS

If you installed batch processing as a daemon using the Configuration Assistant, you can stop it and restart it using the `launchctl` command line tool. If you installed it using Service Launcher, you can stop it and restart it manually in Service Launcher.

How to stop and restart the batch processing daemon


1. In a terminal, enter the following command:

```
$ sudo launchctl unload  
/Library/LaunchDaemons/com.toonboom.QueueServer.plist
```

2. When you are ready to start the server again, enter the following command:

```
$ sudo launchctl load  
/Library/LaunchDaemons/com.toonboom.QueueServer.plist
```

How to stop and restart the batch processing daemon in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the top menu, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **ToonBoom Process** tab.
4. Click on **Stop**.
5. When you are ready to start the server again, click on **Launch**.

Interrupting a Batch Processing Task on macOS

Control Center will allow you to cancel a batch processing task before it starts by deleting it from the queue, but if you delete a batch processing task while it is already processing, it will still finish in the background.

Vectorizing tasks are added to the queue on a per-drawing basis. This means that if you want to interrupt the vectorization of a scene, you can simply stop the batch processing service and clear the queue in Control Center. However, rendering tasks are added to the queue on a per-scene basis. Therefore, the only way to interrupt the rendering of a scene while it is currently rendering is by interrupting the Harmony process that is rendering the scene.



IMPORTANT

You should only interrupt a batch processing task when it is necessary, such as when an extremely lengthy task was started by accident and is liable to delay the other tasks.

How to interrupt a batch processing job

1. In a terminal, enter the following command:

```
$ pgrep tbprocess
```

The command will output the process ID (*pid*) of the batch processing service (**tbprocess**).

2. Enter the following command to get the process ID of the child process of the batch processing service, replacing `<pid>` with the process ID of the batch processing service:

```
$ pgrep -P <pid of tbprocess>
```

This outputs the process ID of the shell in which the batch processing job is being ran.

3. Enter the same command, but this time, put the process ID of the child process of the batch processing service.

```
$ pgrep -P <pid of shell>
```

This outputs the process ID of the Harmony session that is rendering the job.

4. Enter the same command, but again, put the process ID returned by the last command instead:

```
$ pgrep -P <pid of Harmony>
```

This either outputs nothing or, if Harmony is rendering a movie file, the process ID of the media server creating the movie file.

5. Enter the following command followed by the process IDs for the shell, the Harmony session and, if applicable, the utility creating the movie file:

```
$ sudo kill -SIGKILL <pid of shell> <pid of Harmony> <pid of media server>
```

**TIP**

You can execute each command in one go by copying and pasting the following line to the terminal:

```
$ TBPROCESSPID=$(pgrep tbprocess) && TBSHELLPID=$(pgrep -P $TBPROCESSPID) && TBHARMONYPID=$(pgrep -P $TBSHELLPID) && TBEDIAPID=$(pgrep -P $TBHARMONYPID); sudo kill -SIGKILL $TBSHELLPID $TBHARMONYPID $TBEDIAPID
```

**NOTE**

The job you interrupted will still appear in the queue with the **Processing** status in Control Center. You will need to delete the job manually from the queue to clear it up.

Uninstalling the Batch Processing Daemon on macOS

If you installed batch processing as a daemon using the Configuraton Assistant, you can uninstall it using the `launchctl` command line tool. If you installed it using Service Launcher, you can uninstall it in Service Launcher.

How to uninstall the batch processing daemon

1. In a terminal, enter the following command:


```
$ sudo launchctl unload -w  
/Library/LaunchDaemons/com.toonboom.QueueServer.plist
```

The daemon will no longer launch on boot.

2. If you want, you can delete the daemon permanently with the following command:

```
$ sudo rm /Library/LaunchDaemons/com.toonboom.QueueServer.plist
```

How to uninstall the batch processing daemon in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the top menu, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **ToonBoom Process** tab.
4. Click on **Stop**.
5. Uncheck the **Launch when Service Launcher starts** option.
6. In the top menu, select **Server > Save All Settings**.

Installing the Batch Processing Service on GNU/Linux

On GNU/Linux, you can install the batch processing service by enabling its daemon to make it launch on boot as user `usabatch`, or Service Launcher to make it launch when the current user logs in.

Installing the Batch Processing Daemon on GNU/Linux	309
Running the Batch Processing Service on Log-In on GNU/Linux	311
Checking the Batch Processing Log on GNU/Linux	313
Launching Batch Processing Manually on GNU/Linux	314
Stopping and Restarting the Batch Processing Daemon on GNU/Linux	315
Interrupting a Batch Processing Task on GNU/Linux	316
Uninstalling the Batch Processing Daemon on GNU/Linux	317

Installing the Batch Processing Daemon on GNU/Linux

The Harmony installer for GNU/Linux always installs daemon scripts on your machine, but does not enable them by default. Hence, there are two approaches to installing the batch processing daemon:

- By enabling the daemon that was already installed with Harmony using the **chkconfig** command.
- By running the Harmony installation script with the **-p** parameter.

This daemon runs through the user `usabatch`, so this user must be created prior to installing the daemon.



NOTE

The following steps indicate how to make batch processing launch on boot, via the user `usabatch`, if you prefer, you can make the database server launch on log-in as the logged-in user. For more information, see [Running the Batch Processing Service on Log-In on GNU/Linux on page 311](#).

How to create the `usabatch` user on GNU/Linux

1. In a terminal, enter the following command:

```
$ sudo useradd -m usabatch
$ sudo passwd usabatch
```

2. When prompted, enter the password you want to assign to the user `usabatch`, then confirm it.

The user `usabatch` is now created and can be used to run Harmony daemons.

How to enable the batch processing daemon on GNU/Linux

1. In a terminal, enter the following command to enable the WebCC server daemon:

```
$ sudo chkconfig toonboom_process on
```

If the command returns nothing, it succeeded. The WebCC server will launch next time you boot the machine.

2. To launch the WebCC server daemon immediately, run the following command:

```
$ sudo service toonboom_process start
```

The command will report whether it succeeded or failed.

How to enable the batch processing daemon using the Harmony installation script

1. In a terminal, navigate to the directory where you downloaded the installation package.

```
$ cd ~/Downloads
```

2. Extract the package's contents in the directory.

```
$ tar -xvzf ./name-of-package.tar.gz
```

3. Switch to the directory in which the package's contents were extracted.

```
$ cd name-of-package
```

4. Run the installer with the -d parameter:

```
$ sudo ./install -p
```

5. When prompted, confirm that you want to proceed with the installation.

The installation script will enable and start the batch processing daemon. You can test that your machine is ready to perform batch processing job by entering the following command in a terminal, replacing **<host name>** with the name of your machine.

```
$ Status -b -m <host name>
```

Running the Batch Processing Service on Log-In on GNU/Linux

The traditional way of running the batch processing service is by running the server as a Windows service or daemon. This means the service launches in the background as soon as the machine boots, and starts processing the queue even if nobody is logged in to the computer.

Using Service Launcher, it is possible to configure the batch processing service to launch as the currently logged-in user, when they log in. This means that your machine will only process jobs when the user is logged in.

Although this may seem impractical, it may be preferable for security reasons. Harmony services are run under the user **usabatch**, an user with administrator privileges. When using Service Launcher, the logged-in user does not need to have administrator privileges to run services.

Before configuring Service Launcher to run the batch processing service, the windows service that launches it must be disabled, so as to avoid launching two instances of the batch processing service simultaneously.



NOTE

While daemons run regardless of who is logged in and even when no user is logged-in, Service Launcher will only run when the user for which it was specifically configured is logged in. This means that if you want Service Launcher to launch when any user is logged in, you must configure it for each user that has access to the machine.

How to disable the batch processing daemon on GNU/Linux

1. In a terminal, enter the following command:

```
$ sudo service toonboom_process stop
$ sudo chkconfig toonboom_process off
```

The batch processing daemon will stop and will no longer launch on boot.

How to configure the batch processing daemon to launch on user log-in



1. Make sure you are currently signed-in as the user who will run the service.



NOTE

This user will need to be logged in for the service to be running. The user does not need to have administrator privileges.

2. Do one of the following:

- If Service Launcher is not running, in the GNOME top menu, select **Applications > Harmony 17 [Edition] >  Service Launcher**.
- If Service Launcher is already running, click on its icon  in the system tray.

The Service Launcher application starts.

3. In the **Application parameters** section of the **Preferences** tab, make sure that the **Launch on login** option is checked, so that services configured in Service Launcher automatically launch when you log in to your account.
4. Select the **ToonBoom Process** tab.
5. Make sure the **Schedule** option is checked.
6. Click on **Launch** to verify that the batch processing service launches successfully through Service Launcher.
7. Check the **Launch when Service Launcher starts** option.
8. In the top menu, select **Server > Save All Settings**.

From now on, the batch processing service will launch through Service Launcher when you log in. You can test that your machine is ready to perform batch processing job by entering the following command in a terminal, replacing **<host name>** with the name of your machine.

```
$ Status -b -m <host name>
```

Checking the Batch Processing Log on GNU/Linux


In order to troubleshoot problems with the batch processing daemon, you can look through its log. This is done differently depending on whether you installed the database server as a daemon or using Service Launcher.

How to check the log for the batch processing daemon

1. In a terminal, enter the following command:

```
$ less /tmp/tbprocess.log
```

How to check the log for the batch processing daemon in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the system tray, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **ToonBoom Process** tab.
4. In the bottom half, select the **Log** tab.
5. Click on the **View Complete Log** button in the bottom-right corner.

Launching Batch Processing Manually on GNU/Linux

You can start the batch processing service directly from a terminal. This is usually not practical for a production environment, but it can be very useful when troubleshooting problems. You can view the output of the service directly in the terminal in which you launch it as it starts and handles jobs.

How to start the batch processing daemon manually

1. In a terminal, enter the following command:

```
$ /usr/local/ToonBoomAnimation/harmony[Edition]_17/lnx86_64/bin/tbprocess -schedule
```

Stopping and Restarting the Batch Processing Daemon on GNU/Linux

If you installed batch processing as a daemon, you can stop it and restart it using the **service** command line tool. If you installed it using Service Launcher, you can stop it and restart it manually in Service Launcher.

How to stop and restart the batch processing daemon

1. In a terminal, enter the following command:

```
$ sudo service toonboom_process stop
```

2. When you are ready to start the server again, enter the following command:

```
$ sudo service toonboom_process start
```




TIP

You can also restart the service instantly with the following command:

```
$ sudo service toonboom_process restart
```

How to stop and restart the batch processing daemon in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the system tray, click on the Service Launcher  icon.

The Service Launcher application opens.

3. Select the **ToonBoom Process** tab.
4. Click on **Stop**.
5. When you are ready to start the server again, click on **Launch**.

Interrupting a Batch Processing Task on GNU/Linux

Control Center will allow you to cancel a batch processing task before it starts by deleting it from the queue, but if you delete a batch processing task while it is already processing, it will still finish in the background.

Vectorizing tasks are added to the queue on a per-drawing basis. This means that if you want to interrupt the vectorization of a scene, you can simply stop the batch processing service and clear the queue in Control Center. However, rendering tasks are added to the queue on a per-scene basis. Therefore, the only way to interrupt the rendering of a scene while it is currently rendering is by interrupting the Harmony process that is rendering the scene.



IMPORTANT

You should only interrupt a batch processing task when it is necessary, such as when an extremely lengthy task was started by accident and is liable to delay the other tasks.

How to interrupt a batch processing job

1. In a terminal, enter the following command:

```
$ pgrep tbprocess
```

The command will output the process ID (*pid*) of the batch processing service.

2. Enter the following command to kill all processes started by the batch processing service, which will end any current batch processing job. Replace **<pid of tbprocess>** with the process ID of the batch processing service, obtained with the previous command.

```
$ sudo pkill -SIGKILL -P <pid of tbprocess>
```



NOTE

The job you interrupted will still appear in the queue with the **Processing** status in Control Center. You will need to delete the job manually from the queue to clear it up.

Uninstalling the Batch Processing Daemon on GNU/Linux

If you installed batch processing as a daemon, you can uninstall it using the `chkconfig` command line tool. If you installed it using Service Launcher, you can uninstall it in Service Launcher.


How to disable the batch processing daemon

1. In a terminal, type in the following commands to stop, then disable the daemon:

```
$ sudo service toonboom_process stop
$ sudo chkconfig toonboom_process off
```

The batch processing daemon will stop and will no longer launch on boot.

How to uninstall the batch processing daemon in Service Launcher

1. Make sure you are currently signed in as the user running the service.
2. In the system tray, click on the Service Launcher  icon.
The Service Launcher application opens.
3. Select the **ToonBoom Process** tab.
4. Click on **Stop**.
5. Uncheck the **Launch when Service Launcher starts** option.
6. In the top menu, select **Server > Save All Settings**.

Setting Up Default Schedules

When you first set up your Harmony system, it is a good idea to set up a default batch processing schedule. You can modify this schedule later based on the operating needs of your studio.

If you have not created any environments, there will not be any environments for you to start up.

You must start up a default schedule for every environment you create in the Harmony database.



NOTE

Maintaining a limited number of environments will make it easier to set up and manage batch processing queues. For example, you can create four environments, like **test**, **commercials**, and one environment per features or series.

How to define the default batch processing schedule

1. Open a command prompt or Terminal and type:

> **Setdef**

Follow the onscreen instructions to set up the environment for batch processing by selected machines on the Harmony network.

2. At the next prompt, enter the name of the machine on which you want to set up batch processing:

INPUT A SINGLE MACHINE NAME, THE PREFIX OF A MACHINE NAME, OR all



TIP

If you want to create a schedule for all machines in the machine list, enter **all**.

3. At the following prompt, type in **y** if you want to add more machines to the schedule, or **n** if you are done selecting machines to schedule.

DO YOU WANT TO INPUT MORE MACHINES

4. At the next prompt, type in **y** if you want the machines to be able to vectorize scanned drawings, or **n** if you don't.

is vectorize work allowed?

5. At the next prompt, type in **y** if you want the machines to be able to render scenes, or **n** if you don't.

is render work allowed?

6. At the next prompt, you must select the environments for which the selected machines will perform batch processing jobs. The machine will only vectorize or render jobs that come from scenes in the selected environment.

INPUT A SET OF DATABASE ENVIRONMENTS

The prompt will display a list of environments. Each environment is listed as a number followed by a name. In the prompt, you must type in the number of each environment for which this machine should perform batch processing jobs, separated by spaces, like so:

```
> 1 3 12 34
```

7. At the next prompt, determine if you want batch processing to be enabled or disabled for the selected machines and environment. Typically, you would want to enable batch processing by entering **ON**, but this command can also be used to disable batch processing for machines and environments by entering **OFF**.

input ON or OFF

The utility will immediately output the new status of the selected machines, in this format:

Machine	Status	vec	ren	environments	pri	Time of Last Update
machine-name	ON/OFF	*	*	env env env	*	Mon Jan 01 12:00:00 2018



NOTES

- The environments enabled for the machine are listed using a 3 letters abbreviation. The key for each abbreviation is printed out before the status of the machines.
- An asterix under the **vec** column indicates that the machine is enabled for vectorizing.
- An asterix under the **ren** column indicates that the machine is enabled for rendering.
- An asterix under the **pri** column indicate that the machine has priority.

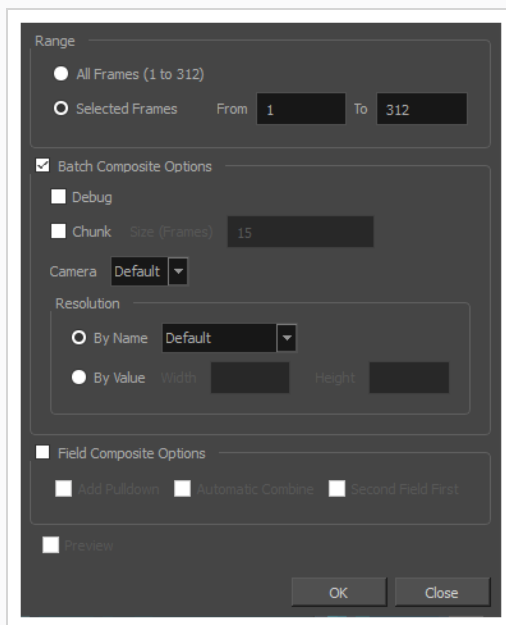
Testing Batch Processing

To verify that batch processing is working, send a few frames from a scene to be rendered by the Harmony module.

How to test batch processing

1. Open the Control Center and select the environment and job that contains the scene to render.
2. Select the scene, right-click on the sample scene in the Scenes section panel and select **Send to Rendering**.

The Render Write Nodes dialog box opens.



3. In the Range section, select **Selected Frames**.
4. In **From** and **To** field, enter a short range of frames, such as 1 to 5.
5. Click on **OK**.
An message appears indicating that the scene was sent to the queue.
6. Click **OK** to close the message .
7. In Control Center, select the environment containing the scene you just sent to the rendering queue.
8. In the top menu, select **Environment > Render Queue**.

The Render Queue dialog opens and displays a list of the scenes in its queue. The scene you sent to rendering should start out with the status **Pending**, then switch to **Processing** and finally to **Completed**.

Vectorization of Scanned Drawings

Before you can paint a scanned drawing in Harmony, you must convert it to a vectorized image format. The vectorizing process converts the drawings from a bitmap format to a vector line-based format.

Manually Vectorizing Scenes or Elements	322
Viewing Drawing Vectorization Lists	323
Modifying Vectorize Queue Entries	325

Manually Vectorizing Scenes or Elements

Normally, when you scan drawings, Harmony automatically vectorizes them (sends them to the Vectorize queue). However, you can also use the Send to Vectorize command to reprocess all drawings for a particular element. Vectorizing prepares images for painting by creating vector files. You would only have to manually send scenes to be vectorized in special cases such as:

- Processing images that were not scanned with the Scan module
- Reprocessing a scene due to unrecoverable paint errors

How to manually vectorize all the drawings for scenes or elements

1. Select a scene from the Scenes list or an element from the Elements list.

If you selected a scene, Harmony vectorizes all drawings for all elements of that scene.

If you selected an element, Harmony vectorizes all its drawings.

2. Depending on the component you selected, do one of the following:

- Select **Scene > Send to Vectorize** or right-click and select **Send to Vectorize**.
- Select **Element > Send to Vectorize** or right-click and select **Send to Vectorize**.

The Confirm dialog box appears.

3. Click **Yes** to vectorize all the selected drawings.

The Control Center module sends all drawings in the scene or element to the Vectorize queue. To verify how the processing is progressing, display the vectorize queue.

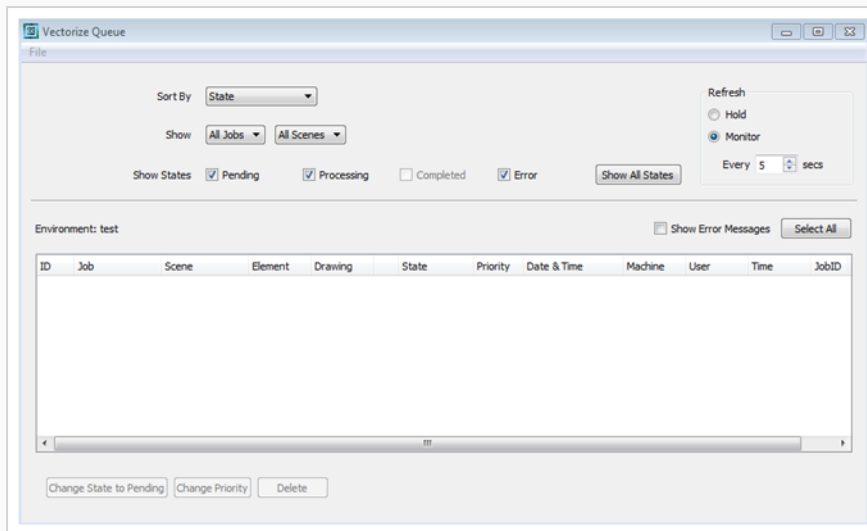
Viewing Drawing Vectorization Lists

The Vectorize queue displays all the drawings that the system is converting to a vector-based format. After the vectorization of a drawing is finished, the completed drawing disappears from the Vectorize queue.

How to view the list of drawings being vectorized

1. Select an environment name from the list, then select **Environment > Vectorize Queue**.

The Vectorize Queue window appears.



2. From the Sort By menu, select how to sort entries in the queue:
 - ▶ **State:** Sorts the entries based on the item's status (Pending, Pending-E, Processing, or Completed)
 - ▶ **Entry time:** Sorts the entries based on the time you send them for vectorization.
 - ▶ **Job, Scene, etc.:** Sorts the entries based on their parent jobs and scenes.
 - ▶ **JobID:** Sorts the entries based on their unique job ID.
3. If the Vectorize queue has too many entries, you can select a specific job, scene, element, or drawing that is in the queue. Select the items you want to display in the queue from the Show menu, which displays all items in the Control Center node's database, not just the ones that have drawings being vectorized.
 - ▶ **Jobs:** Selects the specific job in the queue or one that has scenes currently being vectorized. To see all the jobs, select **All Jobs**.
 - ▶ **Scenes:** Selects the specific scene that is in the queue or currently being vectorized. To see all the scenes in a job, select **All Scenes**.
4. Select the states you want to display in the queue from the Show States options:
 - ▶ **Pending:** Displays the items waiting to be vectorized.

- ▶ **Error:** Displays the items where the system found an error and has stopped vectorizing them (displays Pending-E as the item's state).
 - ▶ **Processing:** Displays the items that the Harmony is currently processing.
 - ▶ **Completed:** Displays the items that were vectorized. However, the Control Center node usually removes completed items from the Vectorize queue, so this option is inactive.
 - ▶ **Show All States:** Displays all the items, regardless of their state.
5. In the Refresh section, select how often you want the queue to check for the latest entries.
- ▶ **Hold:** Displays no new entries to the queue once the button is activated.
 - ▶ **Monitor:** Automatically displays new entries at the interval you enter in the Every ## secs field.
6. To see any related errors for each entry in the queue, select the **Show Error Messages** option.

If the Control Center node detected any errors while it prepares the drawings, it displays Pending-E in the State column. These errors can occur if there are files missing, computers are unavailable, or data cannot be found.

**NOTE**

Most of these errors would occur if the system is improperly configured. Therefore, your system administrator needs to reconfigure the system to correct the error. If you get a Pending-E error, see your system administrator.

Modifying Vectorize Queue Entries

When you have a list of entries in your queue, you can modify them in one of the following ways:

- Click the **Change State to Pending** button to change the status of the selected entries in the queue from Processing to Pending. After a job becomes Pending, the next available machine will process the entry.

If you want to remove the entry from the queue, select the Pending entry and click **Delete**. If the entry is processing, you can delete the entry from the queue but the processing will not stop until it is completed.

You can only modify an entry in the queue while it is pending (check the State column in the queue list). You cannot modify a job that is being processed.

- Click the **Change Priority** button to change the selected drawings' processing priority in the queue. When the New Priority dialog box appears, increase or decrease the drawing's priority.
- Click the **Delete** button to delete the selected drawings from the queue. When you remove a drawing from the queue, the request is cancelled only.

Rendering Scenes

When you have finished working on some scenes, you can send them to the Render queue via Control Center or Harmony, by way of the **Send to Rendering** command. This command allows you to send one or more scenes for rendering to the Batch Processing machine.

You can also use Harmony to render scenes, but since you must open each scene individually in Harmony, you can only send one scene to render at a time.

After rendering a scene, you can mark it as completed using the Change Stage command in the Scene menu. After you render all the scenes for a particular Job, you can mark the Job as completed using the Change Stage command in the Job menu.



NOTE

Changing the job's stage from In Production to Completed only hides the job in the Control Center window.

Viewing Render Queues	327
Modifying Render Queue Entries	329

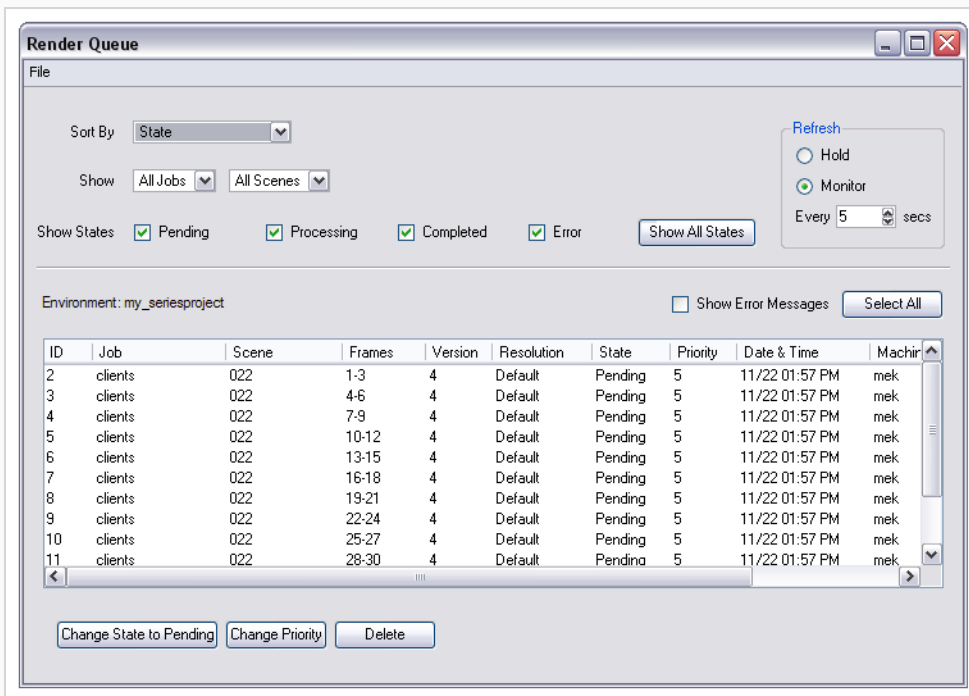
Viewing Render Queues

The Render queue displays all the scenes from the selected environment that the system is rendering. After the system renders a scene, the State of the rendered scene changes to Complete or Completed with Errors and the scene remains in the queue. If you want to remove the rendered scenes, you must select them and use the Delete button.

How to view the scenes being rendered

1. Select an environment name from the list, then select **Environment > Render Queue**.

The Render Queue window appears.



2. Select how to sort the entries in the queue from the Sort By.
 - ▶ **State:** Sorts the entries based on the item's status (Pending, Pending-E, Processing, Completed with Errors, or Completed).
 - ▶ **Entry Time:** Sorts the entries based on the time you sent them for rendering.
 - ▶ **Job, Scene, etc.:** Sorts the entries based on their parent jobs and scenes.
 - ▶ **JobID:** Sorts the entries based on their unique job id.
3. Select the components you want to display in the queue from the Show.
 - ▶ **Jobs:** Select the specific job you want to see.
 - ▶ **Scenes:** Select the specific scene you want to see.
4. Select the states you want to display in the queue from the Show States options.

- ▶ **Pending:** Displays the items waiting in the queue.
 - ▶ **Processing:** Displays the items that currently processing.
 - ▶ **Completed:** Displays completed items.
 - ▶ **Error:** Displays the items that contains errors that stopped processing (displays Pending-E as the item's state).
 - ▶ **Show All States:** Displays all items, regardless of their state.
5. In the Refresh section, select how often you want the queue to check for the latest entries.
- ▶ **Hold:** Displays no new entries to the queue once the button is activated.
 - ▶ **Monitor:** Automatically displays new entries at the interval you enter in the Every ## secs field.
6. Select the **Show Error Messages** option to see any related errors for each entry in the queue.

**NOTE**

If the Control Center node detects any errors while processing the scenes, it displays "Pending-E" in the State column. Errors can occur if there are files missing, computers are unavailable, or data cannot be found.

Many of these errors can occur if the system is improperly configured. If you get a Pending-E error, see your system administrator as the system may need to be reconfigured.

Modifying Render Queue Entries

After you have a list of entries in your queue, you can modify them in one of the following ways:

1. Change their state in the queue. This allows you to update the status of a scene, or set of scenes, in the queue from Processing (which means it is being rendered) to Pending (which means the scenes are waiting to be rendered). After you set a job to Pending, the job remains in the queue until a machine becomes available and takes the job or you delete it from the queue.
 - Click the **Change State to Pending** button to change the status of the selected entries in the queue from Processing (or Completed) to Pending.
2. Change the scene's processing priority. This allows you to change the order in which the system renders the scenes.
 - Click **Change Priority** to change the selected scene's processing priority in the queue.
 - When the New Priority dialog box appears, use the scroll bar to increase or decrease the scene's render priority.
3. Remove scenes from the Render queue. This allows you to cancel the render command on selected scenes.
 - Click the **Delete** button to delete the selected entries from the queue.

You can only modify an entry in the render queue while it is pending (check the State column in the queue list); you cannot modify a job that is being rendered.

If you must change something about the scene's drawings, you must wait until the rendering is complete. Then make your changes and send it to the Render queue.



NOTE

If you absolutely must stop the rendering process, you can kill the render process in Windows, GNU/Linux or macOS.

Batch Processing Schedules

Once the basic vectorization and rendering batch processing is started, there are more advanced options possible such as advanced scheduling.

By creating a schedule to monitor the batch processing, you can determine which machines perform the batch processing and when they do it. Each batch processing machine on your network has its own set of schedules and you can view and modify them from any machine on the network.

There are three types of schedules available for a machine:

- **Supervisory Schedule:** Assigns a one-time batch processing job to a machine. For example, you can define a Supervisory schedule for a machine to perform batch processing work for the next four hours.

Of the three types of schedules, Supervisory schedules have the highest priority. If you define a Supervisory schedule for a specific period of time on a machine, it will perform the batch processing work according to that schedule, regardless of what the Periodic and Default schedules define for that period of time.

- **Periodic Schedule:** Assigns recurring batch processing jobs. For example, you can define a Periodic schedule for a machine to perform batch processing only on Saturdays and Sundays.

Periodic schedules use the days of the week, so you can have batch processing occurring each week on specific days.

In terms of priority, a Periodic schedule supersedes a Default schedule, but not a Supervisory Schedule. If there is a Periodic schedule active on a machine, then it performs this type of batch processing according to its Periodic schedule before it uses the Default schedule.

- **Default Schedule:** Defines which jobs the machine can process if there are no other schedules defined. Default schedules have the lowest priority, and are used only when Supervisory and Periodic schedules are not present or do not define the work to be done for a particular period of time.

When creating or modifying a schedule, you must supply the following information:

- Dates and times for which the schedule applies
- Machine(s) for which the schedule applies
- Whether batch processing is ON or OFF (whether that machine does or does not perform batch processing work)
- The type of batch processing to be done: Vectorize and/or Rendering
- The environments for which batch processing processes the artwork
- The priority of the environments to be batch processed (optional)

For any given machine, you can define, clear and view the Supervisory and Periodic schedules. You can only modify or view the Default schedule for a machine.

You can also shut down and start up environments to stop and start batch processing of a particular environment on a machine(s). When you shut down an environment, you are overriding whatever you already

scheduled for it (according to each type of schedule). Therefore, you will stop any batch processing jobs on the environment until you start that environment again. You can also view a schedule's status to see an overview of all schedules set for a specified time.

Displaying the Schedule Status	332
Reading the Schedule Status List	334
About Default Schedules	336
Displaying Default Schedules	337
Reading Default Schedule Status Lists	339
Modifying Default Schedules	341
About Periodic Schedules	344
Displaying Periodic Schedules	345
Setting Periodic Schedules	347
Clearing Periodic Schedules	351
About Supervisory Schedules	352
Displaying Supervisory Schedules	353
Setting Supervisory Schedules	355
Clearing Supervisory Schedules	359
About Environment Shut Down	360
Shutting Down Environments	361
Starting Up Environments	363
Harmony Scheduling Utilities	365

Displaying the Schedule Status

You can monitor which machines are vectorizing or rendering animation data. The schedule status displays an overview of what each render or vectorize machine is processing at a specific time. The overview also displays a machine's status based on the highest priority schedule set for the time you select.

```

Key:
vec = Vectorize
ren = Render
TES = TEST
Uic = Uictor
dem = demo
dev = dev

pri: indicates if database environments are prioritized,
      or if each environment is processed with equal priority
in R column, R = Running now, at Mon Mar 6 13:44:52 2008
      ? means couldn't determine status
in SD column, * means shutdown is in effect for at least
      some dbenv's, although those may not affect what's scheduled

      SCHEDULE STATUS Mon Mar 6 13:44:00 2008

R SD machine state vec ren environments pri nextevent
R weiner OFF
R hickey OFF
R garneau ON * * Uic dem TES *
R ozone ON *
R jacques OFF
R lincourt OFF
R obrien ON *
R miranda OFF
R pluton ON *
R alexis OFF
R rndtest2 OFF
R demo5 ON * * TES

vectorize: garneau obrien demo5
render: garneau ozone pluton demo5

TEST: garneau demo5
Uictor:
demo:
dev:
  
```

Explains the abbreviations used in the schedule status

Lists the processing status of the batch processing machines in your network

Displays a summary of what each machine is doing

You can display this status list by typing the **status** command in a Command Shell or Terminal on any machine in the network. When you run the **status** command, you must specify the day and time for which you want to view the schedule status.

How to display the schedule status list

1. Open a Command Shell or Terminal and type:

Status

You can view the status for one or more machines by listing them after the **Status** command (instead of all the machines, which is the default). You can also type a prefix to search for machines with names that start with the same prefix.

Example:

- **Status hp**: Shows the status for all machines that begin with "hp".
- **Status cm hp6**: Shows the status for all machines that begin with "cm" and for hp6.
- **Status**: Shows the status for all machines.

When you press Enter/Return, you are prompted to specify the time for which you want to view the schedule status.

INPUT BEGIN TIME in 24-hr. format [15:17]:

4. Type the time you want to view the status for in 24-hour format. If you press Enter/Return without specifying a time, the system uses the time displayed in brackets (the current time).

When you press Enter/Return, the system prompts you for the day for which you want to view the status.

input day [wed] :

5. Type the day of the week for which you want to view the status (you can not enter dates). If you press Enter/Return without entering a day, the system uses the day displayed in brackets (the current day).

The schedule status for the time and day you specified appears.

Reading the Schedule Status List

When you display the Schedule Status list, the information appears in three separate sections:

- **Key:** Lists the abbreviations the status schedule uses for the environments, the two types of batch processes you can schedule on a machine, and the abbreviations the other columns use in the list.
- **Schedule Status:** Displays the processing status of each vectorizing or rendering machine in the network.
- **Summary:** Displays a summary of which machines are performing a specific type of batch processing (vectorize or render). The summary appears directly below the Schedule Status section.

The following table describes the contents of the Schedule Status list:

Column title	Description
R	<p>Displays an R for each machine that has the batch processing service running on it. The batch processing service must be running in order to execute batch processing tasks.</p> <p>If an R does not appear for a machine, the Schedule Status list displays what would be scheduled if the batch processing service was running on it.</p>
SD	<p>Indicates if an environment is shut down for batch processing on that machine. You can specify that certain machines do not process environments by selecting the machines that should be shut down (or turned off).</p> <p>An asterisk (*) appears in the SD column to indicate that one or more environments have been shut down for that machine—see About Environment Shut Down.</p>
Machine	<p>Displays the name of the networked machine that is configured to accept batch processing jobs.</p>
State	<p>Indicates whether the machine is available for batch processing.</p> <p>On: The machine is scheduled to batch process.</p> <p>Off: The machine is not scheduled (for the time you specified) to batch process.</p>
vec status	<p>Lists the type of batch processing the machine is scheduled to perform (vectorize and/or batch processing).</p> <p>If an asterisk (*) appears in these columns for a machine, then this type of batch processing is scheduled for this machine.</p> <p>If a machine is scheduled to do both types of batch processing, the machine performs both types alternatively; it processes an entry from the Vectorize queue,</p>

	vectorizes the data, and then take an entry from the Render queue.
Environments	<p>Displays the environments scheduled to be batch processed on a machine.</p> <p>The machine batch processes only the data from the environments listed in this column.</p>
pri	<p>Displays the priority level for batch processing on a machine. You can prioritize environments so that one machine can perform the batch processing before another.</p> <p>If an asterisk (*) appears in the pri column, the environments are prioritized for that machine. The priority order for the environments appears in the Environments column (reads from left to right, highest to lowest priority).</p>
nexteventTime	<p>Displays the date and time of the next status change for that machine, according to the defined batch processing schedules.</p> <p>For example, even though the hickey machine is off at the moment, the Time column indicates that the next time its status will change is at 12:53 on Wednesday, March 8. This indicates that a Periodic or Supervisory schedule is set for this machine.</p>

About Default Schedules

A Default schedule sets batch processing for a machine when no other schedules are in effect. For example, if there are no Supervisory or Periodic schedules that affect a particular machine on Monday at 9:00 a.m., the Default schedule determines the type of batch processing that this machine performs at that time.

Every Harmony system machine uses a Default schedule to determine when it should perform batch processing. You can display or modify the Default schedules for any machine in the Harmony system.

Displaying Default Schedules	337
Reading Default Schedule Status Lists	339
Modifying Default Schedules	341

Displaying Default Schedules

When you display a Default schedule, you can see all the batch processing machines on the network and when they are scheduled to perform batch processing work. The Default schedule also displays the type of batch processing work (vectorize or render) and which environments are scheduled for the processing work.

After you display the Default schedule, you can modify it to change the time, type of batch processing, or environments to process—see [Modifying Default Schedules](#).

How to display the Default schedule

1. In a GNU/Linux shell, type:

Showdef

You are prompted for the machine(s) whose Default schedule you want to see.

INPUT A SINGLE MACHINE NAME, THE PREFIX OF A MACHINE NAME, OR all [all]:

2. Select the machines you want to view by typing the name of a single machine, a prefix (to select a range of machines that start with the same prefix), or **all** to view all the Default schedules for all the machines.

If you press Enter/Return without specifying a machine, the command uses the machine name that appears in brackets.



NOTE

If you type a prefix, a list of machines that contain that prefix appears. Type the numbers that correspond to the machines you want to view.

For example, if you have three machines that start with "hi", you can type **hi** as a prefix. The system would then display a list of all the machines that start with "hi":

input a set of machines, or hit return to indicate all these machines:

1 = hickey, 2 = highlander, 3=high-definition

You can press Enter/Return to select all the machines in the list or select specific machines by typing their corresponding numbers.

You are asked if you want to view more machines.

DO YOU WANT TO INPUT MORE MACHINES (y/n) [n]:

3. Decide if you want to view more environments and machines.
 - Type **y** to select more machines.
 - Type **n** to not select any more machines.

The Default schedule for the selected machines appears.

The Key lists the abbreviations used in the Default schedule.

```

Key:
vec = Vectorize
ren = Render
tes = test
com = commercials
the = the_show
the = the_movie
tra = training
gro = group_402
pri: indicates if database environments are prioritized,
      or if each environment is processed with equal priority
Note: no time in last column indicates no schedule, and output
      in these cases, if any, shows the default-default settings

```

The Default schedule appears for each machine in your network.

```

DEFAULT SETTINGS Tue Dec 11 10:34:54 2007
Machine Status vec ren environments pri Time of Last Update
mek ON * * tes com the the tra * Tue Dec 11 09:57:02 2007
weiner OFF
hickey OFF
garneau ON * * com the * Wed Dec 8 12:53:00 2007
ozone ON * tra Mon Dec 13 12:34:14 2007
jacques OFF tes
lincourt OFF
obrien ON * Fri Dec 24 12:01:12 2007
miranda OFF
pluton ON * Wed Dec 8 11:56:45 2007
alexis OFF
rndtest2 OFF Wed Dec 8 12:53:00 2007
demo5 ON * * the

```

Reading Default Schedule Status Lists

When you display the Default schedule status list, the system displays the information in two sections:

- **Key:** Lists the abbreviations the status schedule uses for the environments, the two types of batch processes you can schedule on a machine, and the abbreviations the other columns use in the list.
- **Schedule Status:** Displays the processing status of each vectorizing or rendering machine in the network

The following table describes the contents of the Schedule Status list:

Column Title	Description
Machine	Displays the name of the networked machine that is configured to accept batch processing jobs.
Status	Indicates whether the machine is available for batch processing. On: The machine is scheduled to batch process. Off: The machine is not scheduled (for the time you specified) to batch process.
vec ren	Lists the type of batch processing the machine is scheduled to perform (vectorize and/or batch processing). If an asterisk (*) appears in these columns for a machine, then this type of batch processing is scheduled for this machine. If a machine is scheduled to do both types of batch processing, the machine performs both types alternatively; it processes an entry from the Vectorize queue, vectorizes the data, and then take an entry from the Render queue.
Environments	Displays the environments scheduled to be batch processed on a machine. The machine batch processes only the data from the environments listed in this column.
pri	Displays the priority level for batch processing on a machine. You can prioritize environments so that one machine can perform the batch processing before another. If an asterisk (*) appears in the pri column, the environments are prioritized for that machine. The priority order for the environments appears in the Environments column (reads from left to right, highest to lowest priority).
Time of Last Update	The Default schedule also indicates the last time the schedule was modified for each machine.

	This final column displays the date and time that the Default schedule was last modified for each machine.
--	--

Modifying Default Schedules

When you are ready to change the Default schedule, you must specify the following information:

- The machine(s) you want to update
- The type of batch processing work you want to assign
- The environments to process

After modifying the Default schedule, use the **Showdef** command to view your changes—see [Displaying Default Schedules](#).

How to change a Default schedule

1. In a Terminal, Command Prompt or Linux shell, type:

```
Setdef
```

2. Press Enter/Return.

You are prompted for the machine(s) whose Default schedule you want to modify.

```
INPUT A SINGLE MACHINE NAME, THE PREFIX OF A MACHINE NAME, OR all [all]:
```

3. Select the machines you want to view by typing the name of a single machine, a prefix (to select a range of machines that start with the same prefix), or **all** to view all the Default schedules for all the machines.

If you press Enter/Return without specifying a machine, the command uses the machine name that appears in brackets.



NOTE

If you type a prefix, a list of machines that contain that prefix appears. Type the numbers that correspond to the machines you want to view.

For example, if you have three machines that start with "hi", you can type **hi** as a prefix. The system would then display a list of all the machines that start with "hi":

```
input a set of machines, or hit return to indicate all these machines:
```

```
1 = hickey, 2 = highlander, 3=high-definition
```

You can either press Enter/Return to select all the machines in the list or you can select certain machines by typing their corresponding numbers.

You are asked if you want to modify the Default schedule for more machines.

```
DO YOU WANT TO INPUT MORE MACHINES (y/n) [n]:
```

4. Decide if you want to change the Default schedule on other machines.

- Type **y** to select more machines and press Enter/Return.

You are prompted to select other machines (see step 1 of this procedure).

- Type **n** to not select any more machines and press Enter/Return.

You are prompted to decide if you want vectorizing work performed on the machine.

INPUT TYPE OF WORK ALLOWED is vectorize work allowed (y/n) [yes]

5. Decide if you want the machines to perform vectorizing work on the selected machines.
 - Type **y** to schedule vector batch processing and press Enter/Return.
 - Type **n** to not schedule vector batch processing and press Enter/Return.
 - If you press Enter/Return without entering **y** or **n**, the prompt uses the answer shown in brackets.

The system then prompts you to select if you want rendering work performed on the machine.

is render work allowed (y/n) [yes]

6. Select if you want the machines to perform rendering work on the selected machines.
 - Type **y** to schedule Render batch processing and press Enter/Return.
 - Type **n** to not schedule Render batch processing and press Enter/Return.
 - If you press [Enter] without entering **y** or **n**, the prompt uses the answer shown in brackets.

You are prompted to select the environments to be scheduled for batch processing.

INPUT A SET OF DATABASE ENVIRONMENTS

1 = Production, 2 = Tutorials, 3 = Commercials default - [Production Commercials]

All the environments in your system appear in this list, each one assigned to a number.

7. Type the number that corresponds to the environment(s) you want to schedule and press Enter/Return. If you press Enter/Return without typing anything, the environments listed in brackets are used.

If you select more than one environment, the system asks if you want to establish a priority.

should processing of the DBenv's be prioritized? (y/n) [yes]

You can prioritize your environments so that batch processing works on the highest priority environment first. If you do not define a priority, the batch processing does an equal amount of work for each environment.

The order in which you type the environment numbers can determine their batch processing priority in the schedule (the first environment you type has the highest priority). However, the order

in which you type the environments is not important if you do not want to prioritize the environments (by answering **no** to the system prompt).

In this example, if you want to schedule only the **Production** and **Tutorials** environments for batch processing, you would type: **1 2**, indicating that **Production** has a higher priority than **Tutorials** for batch processing.

8. Decide if you want to assign a priority to the environments you selected. If you press Enter/Return without entering anything, the system uses the answer shown in brackets.
 - Type **y** to prioritize batch processing.
 - Type **n** to not prioritize batch processing.

The system prompts you to schedule the status of the machine(s).

input ON or OFF [on]:

9. Decide if you want to apply the Default schedule right away, making the machines available immediately.
 - Type **on** to make the machines available for batch processing according to their defined Default schedules and press Enter/Return.
 - Type **off** to make the machines unavailable for batch processing, regardless of their defined Default schedules, and press Enter/Return.
 - If you press Enter/Return without entering **on** or **off**, the prompt uses the answer shown in brackets.

The Default schedule for the specified machines is set. The new schedule appears.

About Periodic Schedules

The Periodic schedule is an optional schedule that defines a recurring period of time when a machine will perform batch processing. You can create a Periodic schedule by specifying a day of the week (Monday, Tuesday, etc.) and the machine will perform batch processing on the specified days for every week.

A Periodic schedule lists batch processing jobs for which you must define a start and end date, and time. Each job has its own set of batch processing parameters that are independent of the other jobs in the schedule.

For example, you can create a Periodic schedule for a machine so that it performs batch processing every night from 20:00 that evening to 08:00 the following morning. Therefore, the machine would not perform batch processing work during the day when the users would be present.



NOTE

In terms of priority, the Periodic schedule is between the Default and Supervisory schedules. If there are no Supervisory schedules for a machine, or the Supervisory schedule does not define the work to be done for a particular period of time, the machine uses the Periodic schedule.

However, if you have an active Supervisory schedule, the machine follows this schedule before it follows either the Periodic or Default schedules.

Displaying Periodic Schedules	345
Setting Periodic Schedules	347
Clearing Periodic Schedules	351

Displaying Periodic Schedules

When you display a Periodic schedule, you can see all the batch processing machines that use this schedule and when they are scheduled to perform the batch processing. The Periodic schedule also displays the type of batch processing work (vectorize or render) and the environments that are scheduled for the processing work.

After you display the Periodic schedule, you can modify it to change the time, type of batch processing, or environments to process—see [Setting Periodic Schedules](#).



NOTE

You can display the Periodic schedule for only one machine at a time.

How to display a Periodic schedule

1. In a GNU/Linux shell, type:

```
Showper
```

You are prompted for the machine whose Periodic schedule you want to view.

```
INPUT MACHINE [hickey]:
```

2. Type the name of the machine for which you want to define a Periodic schedule.

If you press Enter/Return without specifying a machine, the command uses the machine name that appears in brackets.



NOTE

You can combine these two steps by adding the name of the machine you want to view at the end of the **Showper** command.

For example, if the machine you want to view is called **hickey**, then you would type the following command in a GNU/Linux shell to view its Periodic schedule.

```
Showper hickey
```

The Periodic schedule for that machine appears.

The Key lists the abbreviations used in the Periodic schedule.

The schedule lists when each task starts and stops, including the type of batch processing and the affected environments.

```

PERIODIC SCHEDULE Tue Dec 11 10:50:03 2007
Key:
vec = Vectorize
ren = Render
tes = test
con = commercials
the = the_show
the = the_movie
tra = training
gro = group_402
pri:  indicates if database environments are prioritized,
      or if each environment is processed with equal priority
-----
      Event  vec ren  allowed environments      pri      Time
-----
begin.ON   *  *   tes the tra gro          *      tue 10:48
end.ON
-----
----last updated Tue Dec 11 10:50:03 2007

```

The bottom of the schedule lists the date and time the schedule was last updated for the current machine.

Setting Periodic Schedules

When you are ready to modify your Periodic schedule, you need to specify the following information:

- The machine(s) for which you want to create or update the periodic schedule(s)
- The type of batch processing work you want to assign
- The environments to process
- When to start and stop the batch process

You can add jobs to the schedule if a Periodic schedule already exists for a particular machine, or you can clear out any pre-existing jobs before setting new jobs in the schedule. You can clear a Periodic schedule without setting any new jobs—see [Clearing Periodic Schedules on page 351](#).

How to define a Periodic schedule on a machine



NOTE

If you press Enter/Return without entering anything, the command uses the information that appears in brackets of commands.

1. In a GNU/Linux shell, type:

Setper

You are prompted for the machine(s) whose Periodic schedule you want to modify.

INPUT A SINGLE MACHINE NAME, THE PREFIX OF A MACHINE NAME, OR `all` [`all`]:

2. Select the machines you want to modify by typing the name of a single machine, a prefix (to select a range of machines that start with the same prefix), or **all** to modify all the Periodic schedules for all the machines.



NOTE

If you type a prefix, a list of machines that contain that prefix appears. Type the numbers that correspond to the machines you want to view.

For example, if you have three machines that start with "hi", you can type **hi** as a prefix. The system would then display a list of all the machines that start with "hi":

input a set of machines, or hit return to indicate all these machines:

1 = hickey, 2 = highlander, 3=high-definition

Press Enter/Return to select all the machines in the list or select specific machines by typing their corresponding numbers.

You are asked if you want to modify the Periodic schedule on more machines.

DO YOU WANT TO INPUT MORE MACHINES (y/n) [n]:

3. Do one of the following:
 - Type **y** to select more machines.
 - Type **n** to not select any more machines.

You are prompted to clear the existing schedule before defining new entries.

WOULD YOU LIKE TO CLEAR THE SCHEDULE OF PRE-EXISTING ENTRIES? (y/n) [n]:

4. Do one of the following:
 - Type **y** to remove any existing schedules on the machine(s).
 - Type **n** to preserve any existing schedules on the machine(s). The system will add your new entries to the existing set of entries.

You are prompted to schedule any new entries in your Periodic schedule.

WOULD YOU LIKE TO SCHEDULE ANY ENTRIES? (y/n) [y]

5. Do one of the following:
 - Type **y** to schedule new entries. Proceed to the next step to define the schedule.
 - Type **n** to not schedule any new entries. This cancels the **Setper** command without scheduling any new entries.
6. If you typed **y** to the previous prompt, you are prompted to activate or deactivate the selected machines.

input ON or OFF [on]:

- Type **on** to activate the Periodic schedule for the selected machines.
- Type **off** to deactivate the Periodic schedule for the selected machines. The machine (s) will not perform any Periodic batch processing, regardless of what the other Periodic entries have been scheduled.

You are prompted to decide if you want vectorizing work performed on the machine.

INPUT TYPE OF WORK ALLOWED is vectorize work allowed (y/n) [no]

7. Do one of the following:
 - Type **y** to schedule Vector batch processing.
 - Type **n** to not schedule Vector batch processing.

The system prompts you to decide if you want rendering work performed on the machine.

is render work allowed (y/n) [yes]

8. Do one of the following:

- Type **y** to schedule Render batch processing.
- Type **n** to not schedule Render batch processing.

You are prompted to select the environments to be scheduled for batch processing. All the environments in your system appear in this list, each one assigned to a number.

INPUT A SET OF DATABASE ENVIRONMENTS

1 = Production, 2 = Tutorials, 3 = Commercials default - [Production Commercials]

9. Type the number that corresponds to the environment(s) you want to schedule and press [Enter/Return]. If you select more than one environment, the system asks if you want to establish a priority.

should processing of the DBenv's be prioritized? (y/n) [yes]

You can prioritize your environments so that batch processing works on the highest priority environment first. If you do not define a priority, the batch processing does an equal amount of work for each environment.

The order in which you type the environment numbers can determine their batch processing priority in the schedule (the first environment you type has the highest priority). However, the order in which you type the environments is not important if you don't want to prioritize the environments (by answering **no** to the system prompt).

In this example, if you want to schedule only the **Production** and **Tutorials** environments for batch processing, you would type: **1 2**, indicating that **Production** has a higher priority than **Tutorials** for batch processing.

10. Select if you want to assign a priority to the environments you selected.

- Type **y** to prioritize batch processing.
- Type **n** to not prioritize batch processing.

The scheduler prompts you to define the time and day when to start the Periodic batch processing.

INPUT BEGIN TIME in 24-hr. format [15:17] input day [wed]:

11. Type the time (in 24-hour format) and day (in day of the week format) when to begin batch processing and press [Enter/Return].

The scheduler prompts you for the end time.

INPUT END TIME in 24-hr. format: input day [thu]:

12. Type the time (in 24-hour format) and the day (in day of the week format) when to stop the batch processing and press Enter/Return.

A summary of the job you just scheduled appears. The scheduler then prompts you to confirm that you want to enter this job in the schedule.

```

HERE IS THE EVENT YOU JUST INPUT:
Event   vec ren environments      pri      Time
begin.ON * * tes the tra gro      *      tue 10:48
end.ON

```

13. Do one of the following:

- Type **y** to add this job into your Periodic schedule.
- Type **n** if you do not want to add this job into your Periodic schedule.

The scheduler gives you the opportunity to add another job to your Periodic schedule.

DO YOU WANT TO SCHEDULE ANOTHER EVENT (y/n) [y]

14. Do one of the following:

- Type **y** to add another job and press Enter/Return. Repeat steps 7 to 13.
- Type **n** if you do not want to enter another job and press Enter/Return.

When you finish adding jobs to the Periodic schedule, a summary of the entire Periodic schedule appears.

Clearing Periodic Schedules

If you decide that you do not need to have any recurring batch jobs, you can clear the scheduled jobs from the Periodic schedule. Once the Periodic schedule is empty, the batch jobs in the Default schedule come into effect.

You can clear existing jobs from a Periodic schedule without creating any new jobs. However, when you clear a Periodic schedule, you clear all the jobs that exist for that schedule.

How to clear a Periodic schedule

1. Follow steps 1 to 3 in [Setting Periodic Schedules on page 347](#).
2. Type **y** when you reach the following prompt:

```
WOULD YOU LIKE TO CLEAR THE SCHEDULE OF PRE-EXISTING ENTRIES? (y/n) [n]
```

3. Type **n** when you reach the following prompt:

```
WOULD YOU LIKE TO SCHEDULE ANY ENTRIES? (y/n) [y]
```

This clears the Periodic schedule and displays an empty schedule, confirming that the system cleared the jobs.

```

PERIODIC SCHEDULE Tue Dec 11 11:09:48 2007
Key:
vec = Uectorize
ren = Render
tes = test
com = commercials
the = the_show
the = the_movie
tra = training
gro = group_402
pri: indicates if database environments are prioritized,
      or if each environment is processed with equal priority
-----
      allowed
Event  vec ren  environments      pri      Time
-----
----last updated Tue Dec 11 11:09:48 2007

```


About Supervisory Schedules

A Supervisory schedule forces a machine to perform batch processing for a non-recurring period of time. This type of schedule, like the Periodic schedule, is optional. If you have a Supervisory schedule active for a period of time, the machine performs batch processing according to that schedule before it references the Periodic or Default schedules.

For example, you could force a machine to perform only Vectorize batch processing for the next four hours, ignoring any other scheduled type of batch processing during that time.

A Supervisory schedule consists of a list of batch processing jobs for which you must define a start and end date and time. Each job has its own set of batch processing parameters that are independent of the other jobs in the schedule.



NOTE

The Supervisory schedule holds the highest priority level, superseding the Periodic schedule and Default schedule. Therefore, if you have an active Supervisory schedule, the machine follows this schedule before it follows the Periodic or Default schedules.

Displaying Supervisory Schedules	353
Setting Supervisory Schedules	355
Clearing Supervisory Schedules	359

Displaying Supervisory Schedules

When you display a Supervisory schedule, you can see all the batch processing machines that use this schedule and when they are scheduled to perform the batch processing. The Supervisory schedule also displays the type of batch processing work (vectorize or render) and the environments that are scheduled for the processing work.

After you display the Supervisory schedule, you can modify it to change the time, type of batch processing, or environments to process—see [Setting Supervisory Schedules](#).



NOTE

You can display the Supervisory schedule for only one machine at a time.

How to display a Supervisory schedule

1. In a GNU/Linux shell, type:

Showsup

You are prompted for the machine whose Supervisory schedule you want to view.

INPUT MACHINE [hickey]:

2. Type the name of the machine for which you want to define a Supervisory schedule.

If you press Enter/Return without specifying a machine, the command uses the machine name that appears in brackets.



NOTE

You can combine these two steps by adding the name of the machine you want to view at the end of the **Showsup** command.

For example, if the machine you want to view is called **hickey**, then you would type the following command in a GNU/Linux shell to view its Supervisory schedule.

Showsup hickey

The Supervisory schedule for the machine appears.

The Key lists the abbreviations used in the Supervisory schedule.

```

SUPERVISORY SCHEDULE Tue Dec 11 11:15:22 2007

Key:
vec = Vectorize
ren = Render
tes = test
com = commercials
the = the_show
the = the_movie
tra = training
gro = group_402
pri:  indicates if database environments are prioritized,
      or if each environment is processed with equal priority

```

The schedule lists when each task starts and stops, including the type of batch processing and the affected environments.

```

-----
Event  vec ren  allowed environments      pri      Time
-----
begin.ON *  *   tes the gro              *      Tue Dec 11 21:00:00 2007
end.ON
begin.ON *  *   tes com the tra         *      Wed Dec 12 07:00:00 2007
end.ON
begin.ON *  *   tes com the tra         *      Wed Dec 12 15:00:00 2007
end.ON
-----
----last updated Tue Dec 11 11:15:22 2007

```

The bottom of the schedule lists the date and time the schedule was last updated for the current machine.

Setting Supervisory Schedules

When you are ready to modify your Supervisory Schedule, you need to specify the following:

- The machine(s) for which you want to create or update the supervisory schedule(s)
- The type of batch processing work you want to assign
- The environments to process
- When to start and stop the batch process

You can add jobs to the schedule if a Supervisory Schedule already exists for a particular machine, or you can clear out any pre-existing jobs before setting new jobs in the schedule. You can clear a Supervisory Schedule without setting any new jobs (see [Clearing Supervisory Schedules on page 359](#)).

How to define a Supervisory schedule on a machine



NOTE

If you press Enter/Return without entering anything, the command uses the information that appears in brackets of commands.

1. In a GNU/Linux shell, type:

Showsup

You are prompted for the machine(s) whose Supervisory schedule you want to modify.

INPUT A SINGLE MACHINE NAME, THE PREFIX OF A MACHINE NAME, OR `all` [`all`]:

2. Select the machines you want to modify by typing the name of a single machine, a prefix (to select a range of machines that start with the same prefix), or **all** to modify all the Supervisory schedules for all the machines.



NOTE

If you type a prefix, a list of machines that contain that prefix appears. Type the numbers that correspond to the machines you want to view.

For example, if you have three machines that start with "hi", you can type **hi** as a prefix. The system would then display a list of all the machines that start with "hi":

input a set of machines, or hit return to indicate all these machines:

1 = hickey, 2 = highlander, 3=high-definition

Press Enter/Return to select all the machines in the list or select specific machines by typing their corresponding numbers.

You are asked if you want to modify the Supervisory schedule on more machines.

DO YOU WANT TO INPUT MORE MACHINES (y/n) [n]:

3. Do one of the following:
 - Type **y** to select more machines.
 - Type **n** to not select any more machines.

You are prompted to clear the existing schedule before defining new entries.

WOULD YOU LIKE TO CLEAR THE SCHEDULE OF PRE-EXISTING ENTRIES? (y/n) [n]:

4. Do one of the following:
 - Type **y** to remove any existing schedules on the machine(s).
 - Type **n** to preserve any existing schedules on the machine(s). The system will add your new entries to the existing set of entries.

You are prompted to schedule any new entries in your Supervisory schedule.

WOULD YOU LIKE TO SCHEDULE ANY ENTRIES? (y/n) [y]

5. Do one of the following:
 - Type **y** to schedule new entries. Proceed to the next step to define the schedule.
 - Type **n** to not schedule any new entries. This cancels the **Setper** command without scheduling any new entries.
6. If you typed **y** to the previous prompt, you are prompted to activate or deactivate the selected machines.

input ON or OFF [on]:

- Type **on** to activate the Supervisory schedule for the selected machines.
- Type **off** to deactivate the Supervisory schedule for the selected machines. The machine(s) will not perform any Supervisory batch processing, regardless of what the other Supervisory entries have been scheduled.

You are prompted to decide if you want vectorizing work performed on the machine.

INPUT TYPE OF WORK ALLOWED is vectorize work allowed (y/n) [no]

7. Do one of the following:
 - Type **y** to schedule Vector batch processing.
 - Type **n** to not schedule Vector batch processing.

The system prompts you to decide if you want rendering work performed on the machine.

is render work allowed (y/n) [yes]

8. Do one of the following:

- Type **y** to schedule Render batch processing.
- Type **n** to not schedule Render batch processing.

You are prompted to select the environments to be scheduled for batch processing. All the environments in your system appear in this list, each one assigned to a number.

INPUT A SET OF DATABASE ENVIRONMENTS

1 = Production, 2 = Tutorials, 3 = Commercials default - [Production Commercials]

9. Type the number that corresponds to the environment(s) you want to schedule and press [Enter/Return]. If you select more than one environment, the system asks if you want to establish a priority.

should processing of the DBenv's be prioritized? (y/n) [yes]

You can prioritize your environments so that batch processing works on the highest priority environment first. If you do not define a priority, the batch processing does an equal amount of work for each environment.

The order in which you type the environment numbers can determine their batch processing priority in the schedule (the first environment you type has the highest priority). However, the order in which you type the environments is not important if you don't want to prioritize the environments (by answering **no** to the system prompt).

In this example, if you want to schedule only the **Production** and **Tutorials** environments for batch processing, you would type: **1 2**, indicating that **Production** has a higher priority than **Tutorials** for batch processing.

10. Select if you want to assign a priority to the environments you selected.

- Type **y** to prioritize batch processing.
- Type **n** to not prioritize batch processing.

The scheduler prompts you to define the time and day when to start the Supervisory batch processing.

INPUT BEGIN TIME in 24-hr. format [15:17] input day [wed]:

11. Type the time (in 24-hour format) and day (in day of the week format) when to begin batch processing and press Enter/Return.

The scheduler prompts you for the end time.

INPUT END TIME in 24-hr. format: input day [thu]:

12. Type the time (in 24-hour format) and the day (in day of the week format) when to stop the batch processing and press Enter/Return.

A summary of the job you just scheduled appears. The scheduler then prompts you to confirm that you want to enter this job in the schedule.

```

HERE IS THE EVENT YOU JUST INPUT:
Event      vec ren environments      pri      Time
begin.ON   *  *   tes the gro      *      Tue Dec 11 21:00:00 2007
end.ON
  
```

13. Do one of the following:

- Type **y** to add this job into your Supervisory schedule.
- Type **n** if you do not want to add this job into your Supervisory schedule.

The scheduler gives you the opportunity to add another job to your Supervisory schedule.

DO YOU WANT TO SCHEDULE ANOTHER EVENT (y/n) [y]

14. Do one of the following:

- Type **y** to add another job and press Enter/Return. Repeat steps 7 to 13.
- Type **n** if you do not want to enter another job and press Enter/Return.

When you finish adding jobs to the Supervisory schedule, a summary of the entire Supervisory schedule appears.

Clearing Supervisory Schedules

If you decide you don't need to set a Supervisory schedule for your batch processing jobs, you can clear the scheduled jobs from the Supervisory schedule. Once the Supervisory schedule is empty, the batch processing jobs in the Periodic and Default schedules come into effect.

You can clear existing jobs from a Supervisory schedule without creating any new jobs. However, when you clear a Supervisory schedule, you clear all of the jobs that exist for that schedule.

How to clear a Supervisory schedule

1. Follow steps 1 to 3 in [Setting Supervisory Schedules on page 355](#).
2. Type **y** when you reach the following prompt:

```
WOULD YOU LIKE TO CLEAR THE SCHEDULE OF PRE-EXISTING ENTRIES? (y/n) [n]
```

The Supervisory schedule is cleared. An empty schedule displayed, confirming that the system cleared the jobs.

```

SUPERVISORY SCHEDULE Tue Dec 11 11:22:10 2007
Key:
vec = Uectorize
ren = Render
tes = test
con = commercials
the = the_show
the = the_movie
tra = training
gro = group_402
pri: indicates if database environments are prioritized,
      or if each environment is processed with equal priority
-----
Event  vec ren  allowed environments      pri      Time
-----
----last updated Tue Dec 11 11:22:10 2007

```


About Environment Shut Down

In some cases, you must shut down a machine so it does not process any batch jobs from an environment or for a specific amount of time. You may want to, for example, shut down batch processing for all environments on a single machine so you can reboot the machine. Or you can shut down one environment on all machines so you can move scenes in that environment to other machines.

To determine if an environment is shut down on a particular machine, type **Status** in a Command Shell or Terminal. If you see an asterisk (*) in the **SD** column for a particular machine, this means at least one environment is shut down on that machine.

You can view the Supervisory and Periodic schedules for the shut down machines to isolate which environments are shut down.

Shutting Down Environments	361
Starting Up Environments	363

Shutting Down Environments

When you shut down an environment, this overrides whatever was scheduled for that environment (for all three types of schedules). The environment remains shut down until you reinstate it by typing **Suenv** command in a Command Shell or Terminal (see [Starting Up Environments](#)). You can shut down multiple environments on multiple machines, from any machine in the network.

When shutting down an environment, the command prompts you for the environments you want to close and the machines on which you want them to be shut down.

How to shut down an environment on a machine

1. In a Command Shell or Terminal, type:

```
Sdenv
```

You are prompted for the environments to shut down:

```
INPUT DATABASES TO BE STOPPED
```

```
INPUT A SET OF DATABASE Environments
```

```
1 = Production, 2 = Tutorials, 3 = Commercials, default = [Production  
Tutorials Commercials]
```

All the Environments appear in this list and each one has its own number.

2. Type the number that corresponds to each environment you want to shut down (separate each number by a space) and press Enter/Return.

You are prompted you for the machine(s) on which you want the environments to be shut down:

```
INPUT MACHINE ON WHICH THESE DBENV'S ARE TO BE SHUTDOWN
```

```
INPUT A SINGLE MACHINE NAME, THE PREFIX OF A MACHINE NAME, OR all  
[hickey]:
```

3. Select the machines that should not process the shut down environment by typing the name of a single machine, a prefix (to select a range of machines that start with the same prefix), or **all** to shut down all the environments on all the machines.

If you press Enter/Return without specifying a machine, the command uses the machine name that appears in brackets.

**NOTE**

If you type a prefix, a list of machines that contain that prefix appears. Type the numbers that correspond to the machines you want to view.

For example, if you have three machines that start with "hi", you can type **hi** as a prefix. The system would then display a list of all the machines that start with "hi":

input a set of machines, or hit return to indicate all these machines:

1 = hickey, 2 = highlander, 3=high-definition

Press Enter/Return to select all the machines in the list or select specific machines by typing their corresponding numbers.

You are prompted to decide if you want to shut down more machines.

DO YOU WANT TO INPUT MORE MACHINES (y/n) [n]:

You are asked if you want to shut down more environments and machines.

4. Do one of the following:

- Type **y** to select more machines.
- Type **n** to not select any more machines.

A summary of the shutdown appears:

here are the 3 machines you input: hickey weiner garneau

ALL MACHINES HAVE REPORTED. SHUT DOWN COMPLETE.

The **Sdenv** command waits for a response from all the selected machines. The **Sdenv** command does not terminate a batch process that is currently running on a machine. If a machine is currently performing a batch process, it will not shut down until it completes the queue entry.

Starting Up Environments

When you're ready to restart the batch process on an Environment that was shut down, use the **Suenv** command. When you restart an Environment, you are prompted for the Environments you want to start up and the machines you want them to start running on.



NOTE

If you accidentally restart an environment that is already active, nothing happens.

How to start an environment

1. In a Command Shell or Terminal, type:

```
Suenv
```

You are prompted for the environments to restart:

```
INPUT DATABASES TO BE STARTEDINPUT A SET OF DATABASE Environments  
1 = Production, 2 = Tutorials, 3 = Commercials, default = []
```

All the environments in your system appear in this list and each one has a number.

2. Type the numbers that correspond to each environment you want to restart (separate each number by a space) and press Enter/Return.

You are prompted for the machine(s) you want to process the environments:

```
INPUT MACHINE ON WHICH THESE DBENV'S ARE TO BE STARTED  
INPUT A SINGLE MACHINE NAME, THE PREFIX OF A MACHINE NAME, OR all  
[hickey]:
```

3. Select the machines for processing the environment by typing the name of a single machine, a prefix (to select a range of machines that start with the same prefix), or **all** to restart all the environments on all the machines.

If you press Enter/Return without specifying a machine, the command uses the machine name that appears in brackets.

**NOTE**

If you type a prefix, a list of machines that contain that prefix appears. Type the numbers that correspond to the machines you want to view.

For example, if you have three machines that start with "hi", you can type **hi** as a prefix. The system would then display a list of all the machines that start with "hi":

```
input a set of machines, or hit return to indicate all these machines:
```

```
1 = hickey, 2 = highlander, 3=high-definition
```

Press Enter/Return to select all the machines in the list or select specific machines by typing their corresponding numbers.

You are prompted to decide if you want to restart environments on more machines.

```
DO YOU WANT TO INPUT MORE MACHINES (y/n) [n]:
```

4. Do one of the following:

- Type **y** to select more machines.
- Type **n** to not select any more machines.

A summary of the started machines appears:

```
here are the 3 machines you input: hickey weiner garneau  
ALL MACHINES HAVE REPORTED. STARTUP COMPLETE.
```

Harmony Scheduling Utilities

The following table lists the utilities used by Harmony to configure batch processing schedules and their parameters.

Command	Effect
Status	Displays the Schedule Status (default for all machines).
	Options: [machine]: Limits the display to specified machines (separate machine names with spaces). [prefix]: Limits the display to machines with specified prefix.
Showdef	Displays Default schedules.
Setdef	Modifies Default schedules.
Showper	Displays Periodic schedule (for a single machine only).
	Options: [machine]: Specifies the machine whose Periodic schedule you want to display.
Setper	Sets and/or clears Periodic schedule.
Showsup	Displays Supervisory schedule (for a single machine only).
	Options: [machine]: Specifies the machine whose Supervisory schedule you want to display.
Setup	Sets and/or clears Supervisory schedules.
Sdenv	Shuts down environments.
Suenv	Starts up environments.

Chapter 5: Service Launcher

The Service Launcher allows users without full administrative rights to launch three Toon Boom Harmony services: ToonBoom Process (for batch rendering and vectorization), Database Server and WebCC. It will also attempt relaunch services should they stop for any reason.

The topics in this installation guide specify how to perform many tasks with Service Launcher. The following section is a reference guide for the user interface of Service Launcher.

Service Launcher Top Menu	367
Service Launcher Preferences Tab	368
Service Launcher ToonBoom Process Tab	369
Service Launcher Database Server Tab	371
Service Launcher WebCC Tab	373
Service Launcher Link Tab	375

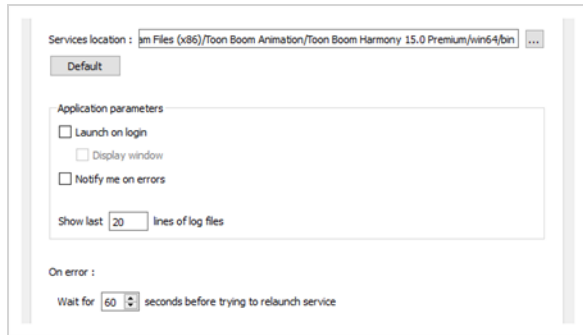
Service Launcher Top Menu

Access these menus and their commands from the top menu bar of the Service Launcher window.

Menu	Command	Description
Server	Save All Settings	Select this command to register any changes made to the options in the Service Launcher.
	Stop All Processes	Select this command to stop all service processes.
	Quit	Select this command to quit the Service Launcher application.
Help	Help	Select this command to launch Toon Boom's website, where you can find various resources for support.
	About	Select this command to open the About Toon Boom Service Launcher window. This window contains version and copyright information.

Service Launcher Preferences Tab

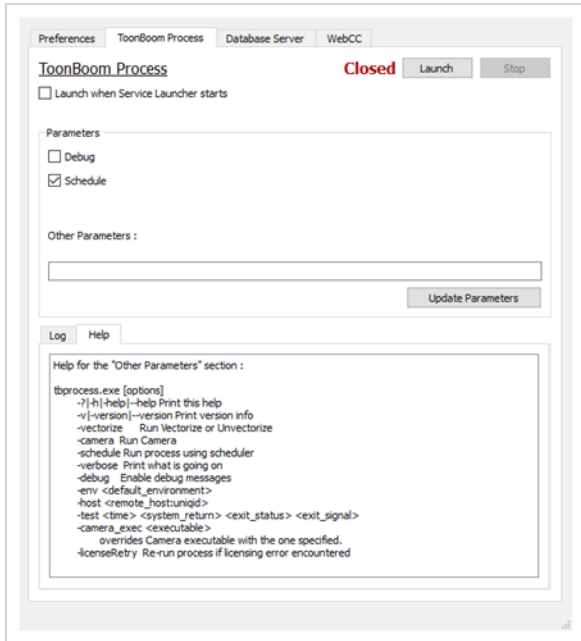
Use the options available in the Preferences tab to customize the behaviour of the Service Launcher.



Parameter	Description
Services location	Displays the location path of the Service Launcher application on your computer. You can click on the ellipses button, at the end of the field, to browse for a new location.
Application parameters	
Launch on login	Check this option if you would like to launch the Service Launcher application every time you login to your computer. This enables the Display Window option.
Display window	Check this option if you would like to the Service Launcher window to open when the Service Launcher is launched on login. Otherwise, the Service Launcher application will run in the background and be accessible to open through your computer's task bar.
Notify me on errors	Check this option if you would like to be notified of errors.
Show last lines of log files	Allows you to enter the maximum number of lines from the bottom of the log file that you would like displayed for the ToonBoom Process, Database Server and WebCC services.
On error	Allows you to enter the number of seconds that the Service Launcher should wait before trying to relaunch any of services, should they happen to stop.

Service Launcher ToonBoom Process Tab

The Toon Boom Process allows a machine to be a batch processing node for your network's vectorizing and render farm. Process runs in the background and checks the database's queue for any vectorizing or rendering job it could perform and, if it find any, performs them in the background.

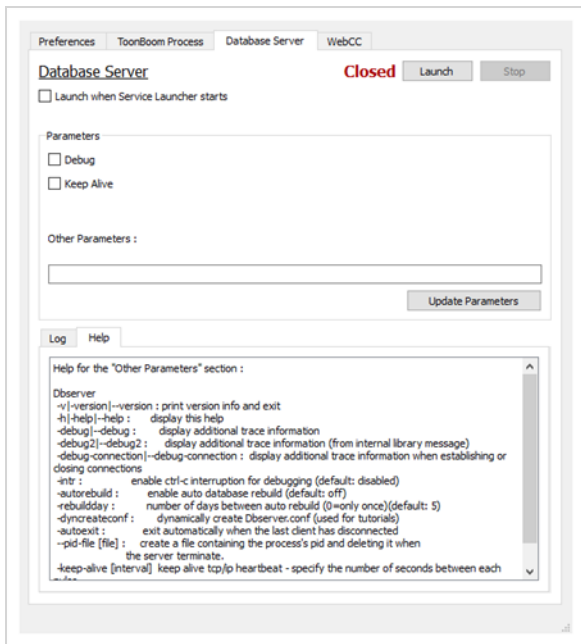


Parameter	Description
Launch button	Click on this button to start the service. The status will change from Closed to Running .
Stop button	This button is enabled when the service is running. Click on this button to stop the service.
Launch when Service Launcher starts	Check this option to automatically launch the ToonBoom Proces service when the Service Launcher is launched. This bypasses the need to click on the Launch button.
Parameters	
Debug	Output detailed information about the tasks performed by the process service to the log, to help debug any issue with the service.
Schedule	This option is checked by default. It is strongly recommended that you leave this option enabled. It runs the ToonBoom Process service using the scheduler.

Other Parameters	Command line parameters to pass to the service when launching it. The Help tab below contains a list of parameters supported by the service. Once you have made changes to the list of parameters, you can click on Update Parameters to relaunch the service with the new parameters.
Help tab	This tab lists all the available options for the ToonBoom Process (tbprocess). You can type these parameters in the Other Parameters field to customize the behaviour of the service.
Log tab	This tab records the activity of the service while it runs. It only lists up to the maximum number of lines specified in the Preferences tab. <ul data-bbox="613 646 1435 762" style="list-style-type: none">• Clear Window button: clears the existing log data from the window.• View Complete Log button: launches the Dialog window which displays a complete version of the log information.

Service Launcher Database Server Tab

The Database Server service manages requests from Harmony Server clients who connect to the Harmony database using Harmony, Paint, Scan or Control Center.

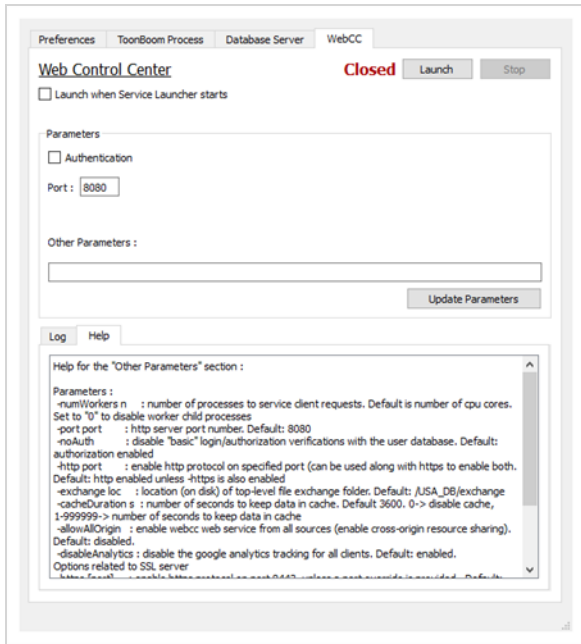


Parameter	Description
Launch button	Click on this button to start the service. The status will change from Closed to Running .
Stop button	This button is enabled when the service is running. Click on this button to stop the service.
Launch when Service Launcher starts	Check this option to automatically launch the Database Server service when the Service Launcher is launched. This bypasses the need to click on the Launch button.
Parameters	
Debug	Output detailed information about the tasks performed by the database server to the log, to help debug any issue with the server.
Keep Alive	Check this option if you would like the Database Server to automatically restart in the event that it crashes or fails. You can select the wait period before relaunch in the Preferences tab.

Other Parameters	Command line parameters to pass to the service when launching it. The Help tab below contains a list of parameters supported by the service. Once you have made changes to the list of parameters, you can click on Update Parameters to relaunch the service with the new parameters.
Update Parameters	Click on this after making changes to the Other Parameters field to stop and relaunch the service with the new parameters.
Help tab	This tab lists all the available options for the database server. You can type these parameters in the Other Parameters field to customize the behaviour of the service.
Log tab	This tab records the activity of the service while it runs. It only lists up to the maximum number of lines specified in the Preferences tab. <ul data-bbox="613 772 1435 884" style="list-style-type: none">• Clear Window button: clears the existing log data from the window.• View Complete Log button: launches the Dialog window which displays a complete version of the log information.

Service Launcher WebCC Tab

The WebCC server service allows people to connect to the Harmony database using a web browser, to export scenes, work on them using Harmony Stand Alone, then upload their changes back to the database.



Parameter	Description
Launch button	Click on this button to start the service. The status will change from Closed to Running .
Stop button	This button is enabled when the service is running. Click on this button to stop the service.
Launch when Service Launcher starts	Check this option to automatically launch the WebCC service when the Service Launcher is launched. This bypasses the need to click on the Launch button.
Parameters	
Authentication	Check this option to display the Login screen when users connect to WebCC. Users must have a valid username and password.
Port	8080 is the default port number for connection. If it was modified, you can enter your custom port number in its place.
Other Parameters	Command line parameters to pass to the service when launching it. The Help tab

	<p>below contains a list of parameters supported by the service. Once you have made changes to the list of parameters, you can click on Update Parameters to relaunch the service with the new parameters.</p>
Update Parameters	<p>Click on this after making changes to the Other Parameters field to stop and relaunch the service with the new parameters.</p>
Help tab	<p>This tab lists all the available options for the WebCC server. You can type these parameters in the Other Parameters field to customize the behaviour of the service.</p>
Log tab	<p>This tab records the activity of the service while it runs. It only lists up to the maximum number of lines specified in the Preferences tab.</p> <ul style="list-style-type: none">• Clear Window button: clears the existing log data from the window.• View Complete Log button: launches the Dialog window which displays a complete version of the log information.

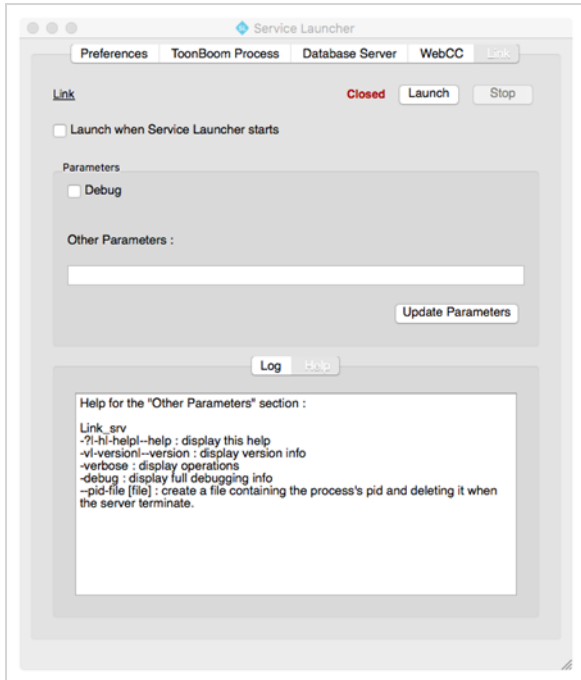
Service Launcher Link Tab

The Link tab allows you to configure the Toon Boom Harmony Link Server to launch when logging-in. The link server allows Windows clients to create symbolic links on macOS or GNU/Linux servers, which is part of the scene creation routine on a Harmony database.



NOTE

This tab is only available on macOS and GNU/Linux.



Parameter	Description
Launch button	Click on this button to start the service. The status will change from Closed to Running .
Stop button	This button is enabled when the service is running. Click on this button to stop the service.
Launch when Service Launcher starts	Check this option to automatically launch the Link server service when the Service Launcher is launched. This bypasses the need to click on the Launch button.
Parameters	
Debug	Output detailed information about the tasks performed by the link server to the

Parameter	Description
	log, to help debug any issue with the server.
Other Parameters	Command line parameters to pass to the service when launching it. The Help tab below contains a list of parameters supported by the service. Once you have made changes to the list of parameters, you can click on Update Parameters to relaunch the service with the new parameters.
Update Parameters	Click on this after making changes to the Other Parameters field to stop and relaunch the service with the new parameters.
Help tab	This tab lists all the available options for the link server. You can type these parameters in the Other Parameters field to customize the behaviour of the service.
Log tab	<p>This tab records the activity of the service while it runs. It only lists up to the maximum number of lines specified in the Preferences tab.</p> <ul style="list-style-type: none"> • Clear Window button: clears the existing log data from the window. • View Complete Log button: launches the Dialog window which displays a complete version of the log information.