Vicon Nexus User Guide

Contents

About this guide

This guide contains instructions for using Vicon Nexus. It explains configuring your Vicon system within Nexus and the basic tasks that make up the everyday Nexus workflow. It assumes you have already installed and licensed Nexus and set up your Vicon system hardware. If you need information about these procedures, see *Installing and licensing Vicon Nexus* and/or the Vicon documentation that was supplied with your hardware, or for help with how to connect up your Vicon system, see Vicon system setup information. You can also contact Vicon Support.

Videos of many of the procedures described in this guide, including many additional tips and examples, are available from the Vicon Nexus 2 Tutorials playlist on YouTube, beginning with system calibration.

> **Note**
>
> As the videos were recorded using an earlier version of Nexus 2, you may notice small differences in the user interface.

About Vicon Nexus documentation

The following documentation is available with Nexus, both as help pages available online and as PDFs that you can download from docs.vicon.com:

Document	Description
What's New in Vicon Nexus	Information about the main features that are new in the current version of Nexus.
Installing and licensing Vicon Nexus	Step-by-step instructions installing and licensing Nexus.
Vicon Nexus User Guide (this guide)	Information about how to use Nexus.
Vicon Nexus Reference Guide	Reference information about less frequently used or more complex procedures, background information to provide you with a better understanding of Nexus, and further details about the Nexus user interface.
Creating labeling skeleton templates (VSTs)	Instructions on how to create your own custom labeling skeleton templates for use with Nexus.
Plug-in Gait Reference Guide	Detailed information on the Plug-in Gait model.

For additional documentation related to Nexus and other Vicon documents, visit docs.vicon.com.

Regulatory information

For Vicon Nexus regulatory details, see Vicon Nexus regulatory information in the Nexus documentation area of the Vicon website (docs.vicon.com).

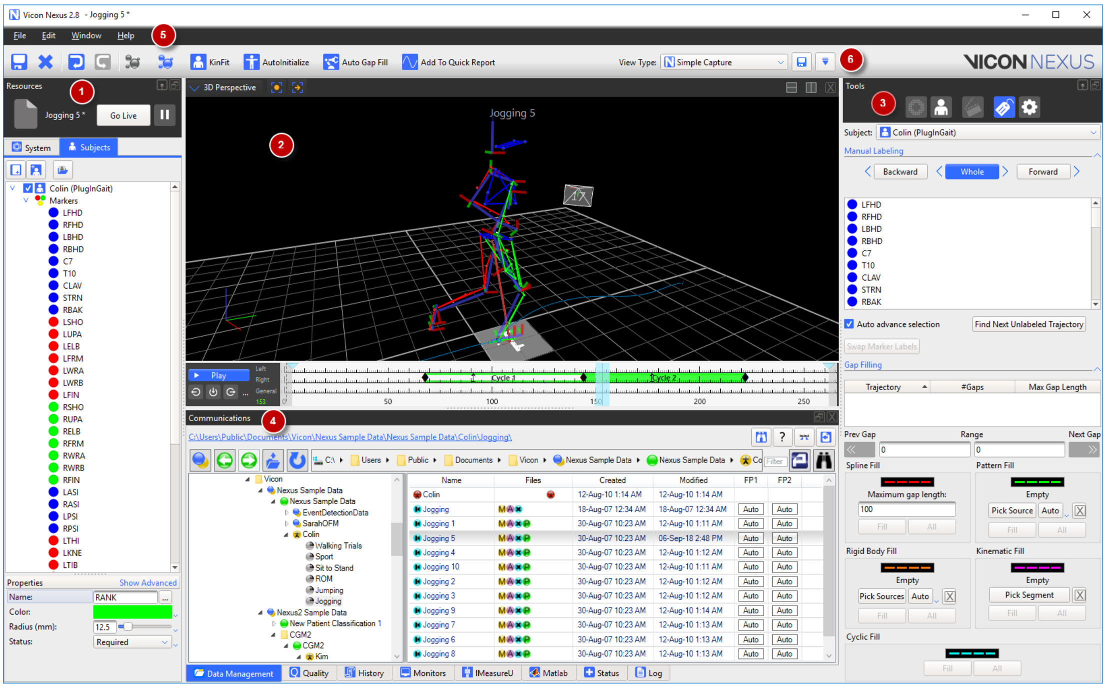
Introducing Vicon Nexus

The following topics will help you to become familiar with the basics of using Nexus:

- Get to know Vicon Nexus on page 8
- Vicon Nexus motion capture workflow on page 29
- Hot keys and shortcuts on page 30

Get to know Vicon Nexus

The Nexus user interface enables you to access the tools you need quickly and easily. Within the primary panes (**Resources**, **View**, **Tools**, and **Communications**), you use the tabs and buttons to open secondary panes containing tools and options for specific parts of the motion capture workflow.



1. **Resources pane**: Manage the different components of your Vicon system, and the subjects whose motion is to be captured, on the **System** tab and the **Subjects** tab. See Manage system and subjects in the Resources pane on page 12.

2. **View pane**: Set up the way you want to visualize the capture data from one or more cameras (or supported third-party devices) either live in real time or offline, from a saved file. See Display data in the View pane on page 16.

3	**Tools pane**: Work through the main stages of the motion capture workflow, using the tabs in the Tools pane from left to right: System Preparation, Subject Preparation, Capture, Label/Edit and Pipeline (this last tab enables you to group and run operations that you use throughout the workflow). See Manage the motion capture workflow in the Tools pane on page 18.
4	**Communications pane:** Store and manage all data associated with your motion capture trials (**Data Management** tab: see Manage motion capture data with the Data Management tab on page 11), assess trial health with the tools (**Quality** tab: see Review data quality on page 154), view stored processing history (History tab: see Review processing history on page 153), set up and control monitors for your trials (**Monitor** tab), work with IMUs on page 222 (**IMeasureU** tab), interact with MATLAB, view system status information (**Status** tab), and view a log of Nexus system activity since start up (**Log** tab).
5	**Menu bar:** Access menu options. See Access menu options from the Nexus menu bar on page 20.
6	**Toolbar:** Access frequently used commands and create and select view types. See Access common commands from the Nexus toolbar on page 20.

Basic keyboard shortcuts and mouse actions

You can use the mouse to manipulate items and manage the way data is visualized in Nexus, and you can combine standard mouse actions with keyboard keys. The following mouse and keyboard combinations are used most frequently in Nexus.

Navigate in the 3D Perspective view:

- Zoom: Right-click + drag forward or backward
- Orbit: Click + drag
- Translate/Move: Click wheel button (or left-and-right-click) + drag

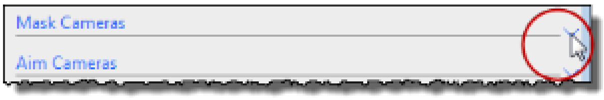
> **Tip**
>
> To open files in Nexus, in addition to loading files as described in Play back data with the time bar on page 19, you can drag and drop Nexus files onto a **3D Perspective** view (or any other view). File types that you can load in this way include: C3D, ENF, VSK/VST, X1D, X2D, and XCP.

Select objects in the view pane:

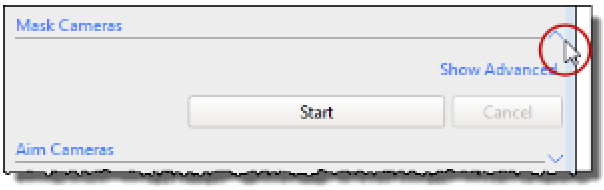
- Select a single item: Click
- Select multiple consecutive items: SHIFT + click
- Select multiple non-consecutive items: CTRL + click
- Select items within a bounding outline: ALT + click and drag

Display/hide a section within a pane:

- Click the Display Section arrow on the right



- Click the Hide Section arrow on the right



For lists summarizing more Nexus shortcuts and mouse actions, see Hot keys and shortcuts on page 30.

Manage motion capture data with the Data Management tab

The **Data Management** tab of the **Communications** window enables you to create a hierarchical structure in which to store and manage all the data associated with your motion capture trials. For information on how to do this, see Prepare a data storage location on page 94.

> **Tip**
>
> To toggle the display of the **Data Management** tab, press F2.

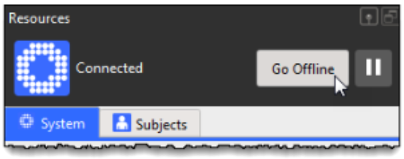
It also enables you to:

- Assess trial health with the tools on the **Quality** tab. For more information, see Review data quality on page 154.
- View processing history on the **History** tab. For more information, see Review processing history on page 153.
- Set up and control monitors for your trials with the **Monitor** tab.
- Interact with MATLAB.
- View system status information on the **Status** tab.
- View a log of Nexus system activity since start up on the **Log** tab.

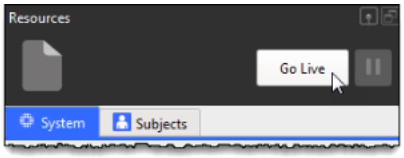
Manage system and subjects in the Resources pane

To manage Nexus system connection and real-time data-streaming, click the system connection buttons at the top of the **Resources** pane:

- **Go Live**: Connect the system and start real-time data streaming. The button's label changes to **Go Offline**.

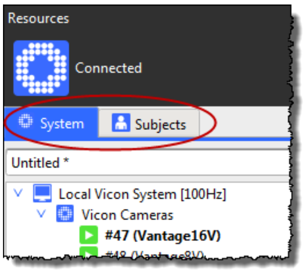


- **Go Offline**: Disconnect the system and stop real-time data streaming. The button's label changes to **Go Live**.



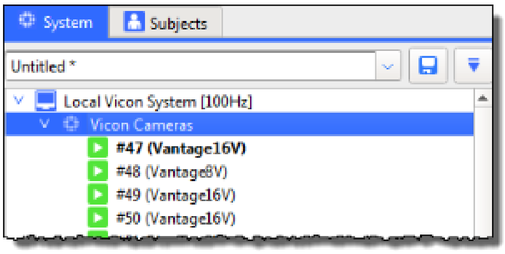
- **Pause**: Pause real-time data streaming. When you click the Pause button while the system is in Live mode, the button turns blue and a pause symbol  is displayed in the view pane. When the system is in Offline mode, the button is unavailable.

At the top of the **Resources** pane, click the tab for the resources you want to manage:



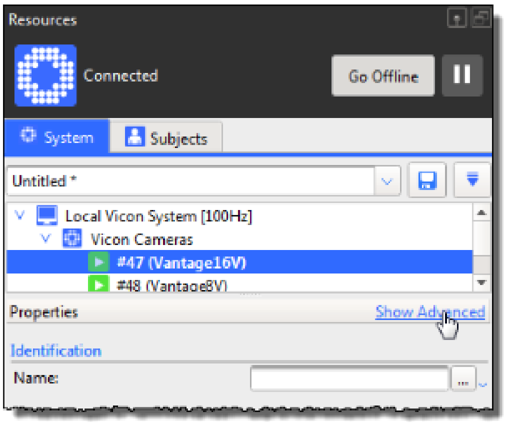
- **System**: View and configure Vicon system components
- **Subjects**: Load and manage files for mocap subjects

In the **System Resources** tree or **Subjects Resources** tree, select the item(s) you want to configure.



Depending on whether you are in Live mode, you can then:

- Right-click a node to display a context (shortcut) menu of commands that can be applied to that item, if one exists.
- In the **Properties** pane below the **Resources** tree, view the settings for the item(s) selected in the tree. To view all of the available properties, click **Show Advanced**.



For more information, see Set properties in Vicon Nexus on page 14.

Set properties in Vicon Nexus

You can configure certain aspects of the Nexus system, such as system components and motion capture subjects, by configuring settings in the corresponding **Properties** section of the user interface.

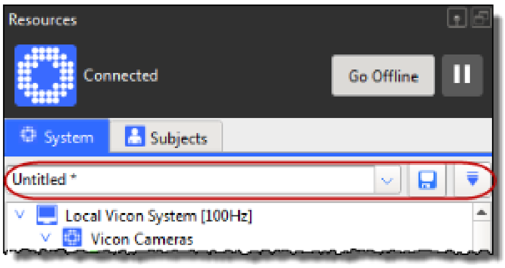
Required properties for which you must specify a value are indicated in the Nexus user interface with a shaded background.

Some properties settings automatically persist, so Nexus remembers them in subsequent sessions. You must explicitly save other settings using the relevant configuration management controls for that area of the Nexus window.

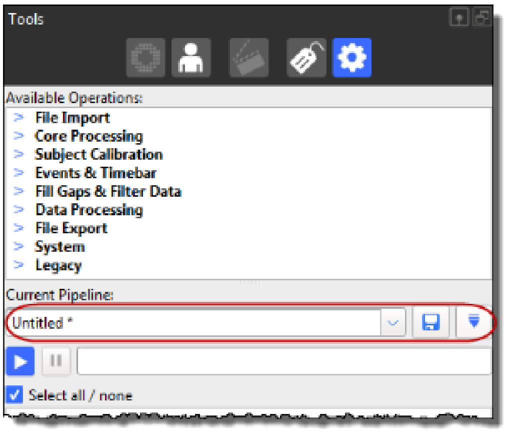
To set properties in Nexus:

1. In the Nexus window, open the pane or dialog box containing the properties whose settings you wish to configure. For example:
   - System components - **System Resources** pane
   - Motion-capture subjects - **Subjects Resources** pane
   - Camera calibration process - **System Preparation Tools** pane
   - Subject calibration process - **Subject Preparation Tools** pane
   - Data processing operations - **PipelineTools** pane
   - Monitor and event actions - **Monitors** tab in the **Communications** pane
   - Data visualization - **Options** dialog box
2. To view all of the available properties, click the **Show Advanced** link. To show only the basic properties, click the **Hide Advanced** link.
3. View or change the setting for the desired properties using its entry field or control.
4. When you are working in the following areas of the Nexus window, you can save any changes you have made to the settings, using the configuration management controls (indicated in the following images). Your settings are saved to a configuration file, so that you can re-use them later.

- **System Resources** pane

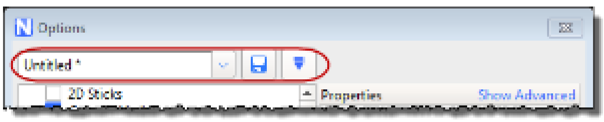


- **Pipeline Tools** pane

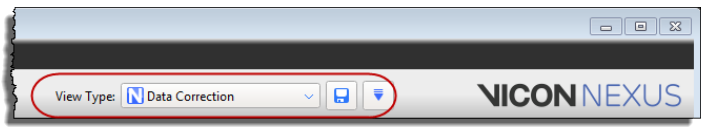


- **Monitors** tab in the **Communications** pane

- **Options** dialog box (press F7 to display)



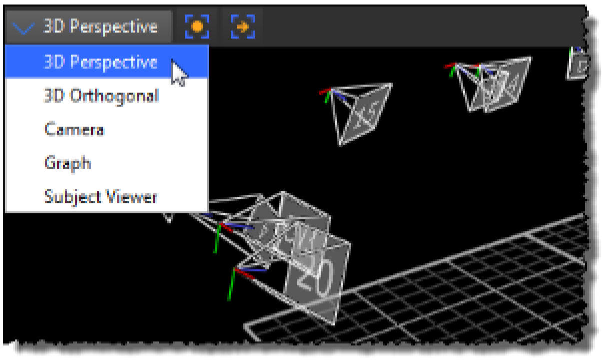
- Nexus toolbar (working with the layout of the view panes)



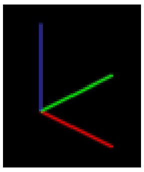
For more information on configuration files, see Manage configurations in Vicon Nexus on page 21.

Display data in the View pane

At the left of the View pane toolbar, click the drop-down list and select one of the available views:

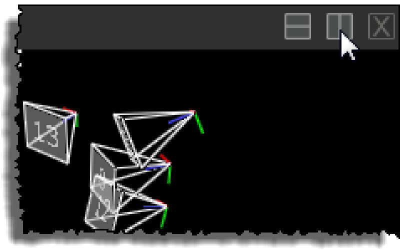


- **3D Perspective**: Display 3D reconstructions of Vicon camera data.
- **3D Orthogonal**: Orthogonal views of 3D data: -Z, +Z, +X, -X, +Y, or -Y

> **Note**
>
>  RGB = XYZ

- **Camera**: Display 2D optical data from Vicon cameras or video streams from connected video cameras.
- **Graph**: Display variables (model outputs), or system components such as force plate or EMG activity.
- **Subject Viewer**: Display the base (default) pose for the labeling skeleton template (VST) of the currently selected subject. This is useful when you are calibrating and manually labeling a labeling skeleton, which are explained in Prepare a subject on page 108.

At the right of the View pane toolbar, click the buttons to specify the number and arrangement of views displayed:



- Horizontal
- Vertical
- Close

Depending on the view selected, additional lists and buttons are available to manage the display options.

Tip

To display multiple **Camera** views, in the **System Resources** tree, SHIFT+click to select multiple cameras and in the View workspace, select **Camera** view.

Manage the motion capture workflow in the Tools pane

At the top of the **Tools** pane, click the buttons for tools relating to the stage in the workflow that you want to display. The buttons are displayed in the order of a typical Nexus workflow (from left to right) and are enabled appropriately, depending on whether you are in Live or Offline mode (the following image is in Live mode).

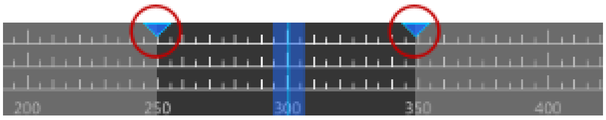
- **System Preparation**: Prepare your Vicon system for motion capture.
- **Subject Preparation**: Prepare subjects whose motion is to be captured.
- **Capture**: Collect motion data.
- **Label/Edit**: Label and fill any gaps in trial data.
- **Pipeline**: Create and manage sequences of operations to process trials.

To find out more about the motion capture workflow, see Vicon Nexus motion capture workflow on page 29.

Play back data with the time bar

To explore the time bar:

1. Ensure a **3D Perspective** view is displayed (see Display data in the View pane on page 16).
2. On the **Data Management** tab at the bottom of the Nexus window, navigate to an existing database. You can do this by expanding the displayed hierarchy and using the **Go forward to the next node**, **Go back to the last node**, and **Move up one folder level** buttons .
3. In the **Name** column, double-click the trial that you want to load.
   The **Data Management** window is minimized and Nexus opens the trial.
4. On the time bar, click the **Play** button or drag the current time indicator (blue vertical line) to play back offline capture data. To crop a trial (restrict playback to a range of frames), you can drag the Start and End Range indicators (the blue triangles at the top of the timeline) along the time bar.

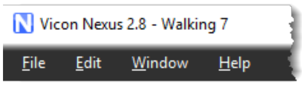


5. To re-display the **Data Management** window, double-click the **Data Management** tab at the bottom of the Nexus window.

> **Tip**
>
> In addition to using the time bar to view and navigate trials, you can also use it to:
>
> - View data quality as a heat map. For more information, see Review trial data using the time bar on page 154.
> - Create and manage events (for example, a foot striking a force plate, or a joint attaining a specified angle, etc). For more information, see Add events to trials on page 180.

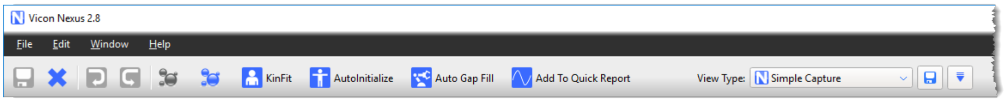
Access menu options from the Nexus menu bar

The Nexus menu bar enables you to access common commands.



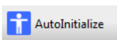
Access common commands from the Nexus toolbar

Access frequently used commands from the Nexus motion capture workflow from the Nexus toolbar.

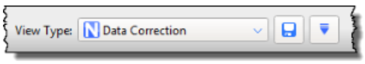


The Nexus toolbar contains buttons that enable you to save the current trial, close the current trial, and undo and redo actions. In addition it contains the following controls:

-  **Reconstruct** Runs the **Reconstruct** pipeline defined in the **Pipeline Tools** pane. (Reconstruction is the process by which Nexus calculates the position of markers in three-dimensional space and links these points frame-by-frame into a trajectory.) For examples of using this pipeline, see Calibrate a labeling skeleton on page 113.
-  **Reconstruct and Label** Runs the **Reconstruct and Label** pipeline defined in the **Pipeline Tools** pane. (The Label process is where labels defined in the labeling skeleton template for the subject are applied, either manually or automatically, to a point in the trajectory of a marker.) Normally used when processing trials. For an example of using this pipeline, see Reconstruct and label movement trials on page 150 .
- 

KinFit Runs the **Kinematic Fit** pipeline defined in the **Pipeline Tools** pane. This pipeline is often used before running a **Fill Gaps - Kinematic** operation for filling gaps in trajectories, and for visualizing or graphing segment- or joint-based data in realtime.
- 

AutoInitialize Runs the **Auto Initialize Labeling** pipeline defined in the **Pipeline Tools** pane. Often used as part of calibrating a labeling skeleton For an example of using this pipeline, see Calibrate a labeling skeleton on page 113.

- Auto Gap Fill **Auto Gap Fill** Runs the **Auto Intelligent Gap Fill** pipeline defined in the **Pipeline Tools** pane. This enables you to quickly fill gaps in your trial, without having to choose which fill method is best for each gap. For good results, you must configure the relevant pipeline operations for your particular trials. For information on using this command, see Automatically fill gaps in trial data on page 177.
- Add To Quick Report **Add To Quick Report** Adds the current trial to a Quick Report. For more information, see Quick Reports in the *Vicon Nexus Reference Guide*.
- **Any user-customized buttons** If required, you can create your own additional buttons and configure them to run a specified pipeline or load a previously created view configuration. You can create or change toolbar buttons in the **Customize Toolbar** dialog box.
- 

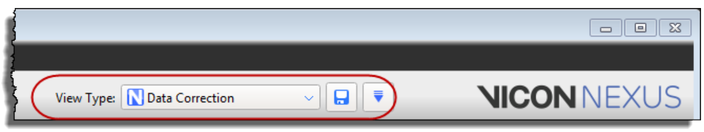
View Type list Lists any saved view types. The adjacent configuration buttons enable you to create and save custom view types.

Clicking a button on the toolbar executes the defined action for the button. A button is dimmed if it is not available, for example, if it cannot be run at that stage of the workflow or if a customized button has been deleted or renamed.

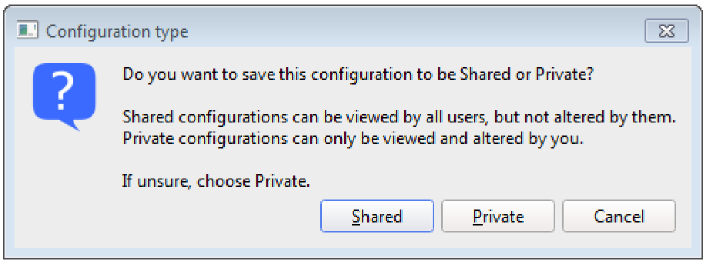
> **Tip**
>
> To display a tooltip that explains why a button is unavailable, hover the mouse pointer over the button.

Manage configurations in Vicon Nexus

You can determine how Nexus looks and behaves by creating and editing configurations in the configuration management controls in the Nexus window. To view and select any default configurations that were installed with Nexus, click the dropdown list in the relevant configuration management controls, for example, for View Types:



You can create different configurations to suit different types of motion capture applications and then select the appropriate configuration when required. You can also save configurations for use by multiple users (Shared) or for a specific user (Private), to suit the way your organization works:



- **Shared** These configurations can be viewed by all users; they can be changed only by the user who was logged on when the file was first created and saved. Shared configuration files are stored in the appropriate folder, by default under: C:\Users\Public\Documents\Vicon\Nexus2.x\Configurations
- **Private** These configurations can be viewed and changed only by the user who was logged on when the file was first created and saved. Private files are stored in the appropriate folder under the logged-in user's Application Data files folder, by default under: C:\Users\<UserName>\AppData\Roaming\Vicon\Nexus2.x\Configurations\
- **System** These configurations are the default configurations that are installed with Nexus and are stored in the Nexus configurations folder, by default: C:\Program Files (x86)\Vicon\Nexus2.#\Configurations

For more information, see Recognize Shared, Private, and System files on page 25.

Important

The default Nexus toolbar is stored in the *Standard.toolbar* configuration file in the **Shared** Nexus configuration folder *Toolbars*. If you add, delete, or reposition buttons on the Nexus toolbar using the **Customize Toolbar** dialog box, these customizations are stored in your **Private** configuration folder.

To manage configurations in Nexus:

1. In the Nexus window, open the pane or dialog box containing the type of Nexus configuration file you want to manage:

Nexus UI area	Configuration type	Configurations folder\file
Capture Tools pane	Motion capture settings	TrialTypes*.TrialTypes
Biomechanics Workflow area of **Communications** pane	Biomechanics workflow	CaptureWorkflows*.CaptureWorkflow
Sounds dialog box	Sounds settings	AudioSchemes*.AudioScheme
Monitors tab in **Communications** pane	Event monitors and actions	Monitors*.Monitors
Options dialog box	Data view options	Options*.Options
Pipeline Tools pane	Automated processing operations	Pipelines*.Pipeline
System Resources pane	System settings	Systems*.System
Toolbar	Toolbar buttons	Toolbars*.Toolbar (see note above)
View pane	View options and layouts	ViewTypes*.ViewType

2. Depending on whether you want to create a new configuration from scratch or change the current configuration or either:
   - Leave the currently loaded configuration file. (If no configuration file has been created yet, Untitled* is displayed and no other options are available.)
     or
   - Select another configuration file from the dropdown list. If you have made changes to the current configuration file, Nexus prompts you to save these before changing the configuration file.
3. In other areas of the Nexus pane or dialog box, make any desired changes to settings, such as those in a Properties section.
4. Click the configuration menu button and select the required command from the displayed list:
   - New: Create a new configuration in which to save the current settings. The name Untitled* is displayed in the Choose configuration list.
   - Save As: In the Save As dialog box, enter a name to overwrite the default new configuration file name Untitled* or to create a new system configuration file in which to save a copy of the current configuration file and click OK.

   Additional options include:
   - Rename: In the Rename dialog box, enter a new name for the currently loaded configuration file and click OK.
   - Delete: At the Delete prompt, click Yes to delete the current file displayed in the configuration dropdown list.
   - Mark Read-Only: Select to protect the current configuration file from further changes.
   - Set Defaults: Returns values to their default settings.
   - Import: Enables you to select and import a configuration file. This is useful when you want to copy a configuration file from another machine.
   - Reload: Reload or delete an automatically saved configuration file. You can select from the list of timestamped files or, if required, delete all the saved configurations.
   - Refresh List: Re-display the contents of the configuration dropdown list. This is useful if you copy a file into one of the Configurations sub-folders, enabling you to update the options displayed in the relevant list, without having to re-start Nexus.

5. In the **Configuration type** dialog box, select the user permissions for the configuration: **Shared** or **Private**.
   The new file name is displayed in the configuration dropdown list.
6. Click the Save button to store the settings in the configuration displayed in the configuration dropdown list.
   

For more information, see Set properties in Vicon Nexus on page 14.

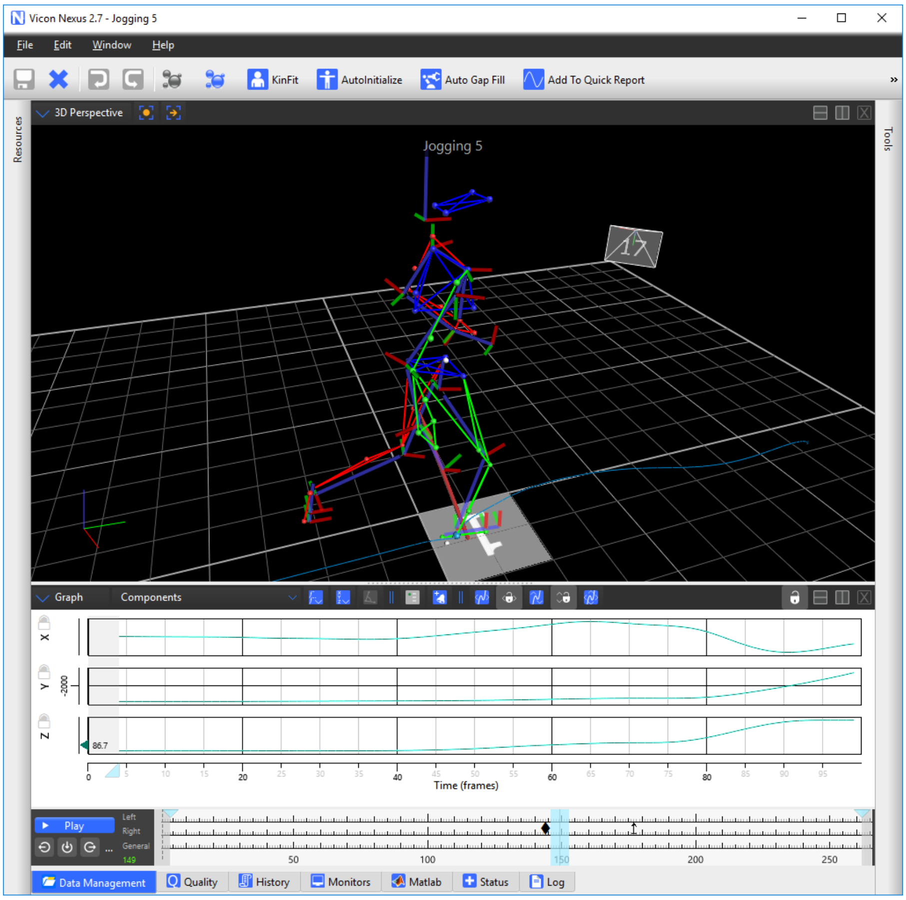
Recognize Shared, Private, and System files

When pipelines, view types, monitors, labeling skeleton templates, and configurations are displayed in Nexus (ie, listed in menus, etc), you can immediately recognize whether they are Private, Shared, or System files:

Icon	File type	Description
	Shared	Can be viewed by multiple users; can be changed only by the user who was logged on when the file was first created and saved. Shared files are stored in subfolders in: C:\Users\Public\Documents\Vicon\Nexus2.x
	Private	Can be viewed and changed only by the user who was logged on when the file was first created and saved. Private files are stored in subfolders in: C:\Users\<*username*>\AppData\Roaming\Vicon\Nexus2.x
	System	Cannot be changed (Read-Only) and are upgraded when the next version of Nexus is installed. Stored in subfolders in the Nexus installation folder, whose default location is: C:\Program Files (x86)\Vicon)\Nexus2.# or C:\Program Files\Vicon\Nexus2.#

Customize the Vicon Nexus user interface

In addition to the usual resize and close window options, you can slide the **Resources** and **Tools** panes to either side of the Vicon Nexus window and or minimize the **Communications** pane, to give you a larger workspace area.



Requirement	Action
To hide the **Resources** and /or **Tools** panes	At the top of the **Resources** or **Tools** pane, click the **UnPin** button. 
To hide the **Communications** pane	Double-click a tab within the **Communications** pane; or To hide the pane whenever you load a trial, on the **Window** menu, select the **Close Communications Pane on Trial Load** option.
To reveal the **Resources** and /or **Tools** panes	Click the tab at the side of the Nexus window  or 
To reveal the **Communications** pane	Double-click a tab within the **Communications** pane; or To make the pane visible whenever you load a trial, on the **Window** menu, clear the **Close Communications Pane on Trial Load** option.
To return a pane to being locked into place	At the top of the pane, click the **Pin** button. 
To undock (float) and dock a pane	At the top of the (pinned) pane, click the **Undock** or **Dock** button.  or 

Customize a View pane

The View pane cannot be undocked, repositioned, or resized in the Nexus window. However, you can open a separate floating view pane by selecting the **New floating workspace** command from the **Window** menu. This floating workspace can be repositioned and resized.

The width of the View pane is affected by resizing panes to the left and/or right of it. The height of the View pane is affected by resizing the panes below it.

Vicon Nexus motion capture workflow

In Vicon Nexus, you can capture and analyze the movement of live subjects (such as human beings or animals) and of inanimate objects (such as sports equipment or other rigid objects) for a variety of motion capture applications. You can either stream motion data in real time or capture it for offline processing, depending on your requirements.

The stages involved in the typical workflow for the operation of Nexus for motion capture production and control are outlined below.

- Prepare a Vicon system on page 43 when you first set up or significantly change your motion capture system.
- Calibrate a Vicon system on page 96 when you first set up your system and regularly afterwards (eg, at the start of every day, before you begin motion capture), to ensure any changes (eg, slight movement of cameras or other equipment) is accounted for. You also do this if you make changes to the system.
- Prepare a subject on page 108 when you have a new subject, or want to make changes to an existing subject. This stage includes calibrating a labeling skeleton for your new subject or re-calibrating when you have made changes to an existing subject.
- Capture movement trials on page 136 after you have calibrated the labeling skeleton, to collect the data needed for your trials.
- Review trials and fill gaps on page 152 after you have captured, reconstructed and labeled your movement trials, to fill any gaps in the data.
- After you have filled any gaps and cropped the data as necessary, you can perform any required modeling (for example, you can run the dynamic Plug-in Gait model, or perform custom modeling) to generate the required model outputs (such as angles, forces, moments, powers, or bones). For information, see Modeling with Plug-in Gait on page 185.
- Export trial data on page 233, which is normally the final stage in motion capture, and enables you to use the captured data in third-party applications.

To speed up some of the above processes, you can use the supplied pipelines, consisting of one or more operations that are supplied with Nexus, or you can create your own custom pipelines. To use pipelines on a large number of trials, you can run them as batch processes. For more information, see Work with pipelines on page 131.

Hot keys and shortcuts

To help you use Vicon Nexus efficiently, this section provides lists of shortcuts:

Common hot keys and shortcuts

You can use the mouse to manipulate items and manage the way data is displayed in the Vicon Nexus window. Standard mouse actions can also be combined with keyboard keys. The following mouse actions and hot keys are available throughout Nexus, where applicable/available:

Task	Keys and mouse
Start/stop capture	CTRL+Enter
Select individual items	Click
Select items within a bounding outline	ALT+click and drag
Select multiple non-consecutive items	CTRL+click
Rotate/orbit	Click and drag

Task	Keys and mouse
Zoom	Right-click and drag
Translate/Move	Click wheel button (or left-and-right-click) and drag
Scroll forward or backward through a list	Rotate mouse wheel
Undo	CTRL+Z
Redo	CTRL+Y
Save currently enabled subject data to the current trial's *.c3d* file (equivalent of clicking **Save** on the **File** menu)	CTRL+S
Reset Core Processor	CTRL+R
Esc	Exit current mode (labeling, etc)
Unset all cameras' Bumped status	CTRL+SHIFT+B

Note

The behavior of the ALT GR key depends upon the regional settings specified for your keyboard in the Windows operating system. In some regions, the behavior of this key is identical to that of the ALT key, while in other regions the ALT GR key functions as if the ALT+CTRL keys were pressed together. Nexus assumes the latter behavior.

If you wish to use the ALT GR key as if it was the ALT key, you must change the regional settings for your keyboard to use the US layout, which assumes identical behavior for these two keys. You change your keyboard language settings in the **Text Services and Input Languages** dialog box, accessed from the **Languages** tab in the **Regional and Language Options** dialog box in Windows **Control Panel**. For more information, see the Microsoft Windows help.

Shortcuts for navigating in Vicon Nexus

Use the following hot keys to navigate to the different areas of the Vicon Nexus user interface.

Task	Keys
Display the Vicon Nexus online help	F1
Display/Close **Data Management** tab	F2
Display/Close the **Quick Reports**window	F4
Enter/Exit full screen mode	F5
Display/Close **Sounds** dialog box	F6
Display/Close **Options** dialog box	F7
Go to **System Preparation Tools** pane	F8
Go to **Subject Preparation Tools** pane	F9
Go to **Capture Tools** pane	F10
Go to **Label/Edit Tools** pane	F11
Go to **Pipeline Tools** pane	F12

The behavior of function keys is dependent upon the area of the Nexus window that has focus when the key is pressed. Click anywhere in the window to set the focus before using the function keys to navigate to a different part of the user interface.

Shortcuts for managing real-time data

Use the following hot keys to manage real-time data streaming and offline data processing in Vicon Nexus.

Task	Keys
Start/stop capture	CTRL+Enter
Switch between Live and Offline mode	CTRL+TAB
Pause/Restart real-time data streaming (Live mode)	SPACE (or middle mouse button)
Play/Stop offline data (Offline mode)	SPACE (or middle mouse button)
Toggle Simple Capture mode (Live mode)	CTRL+H

Shortcuts for selecting items

Use these hot keys and mouse actions to select items in the Vicon Nexus window. To cancel a selection, left-click again in the view pane.

Task	Keys and mouse
Select single item	Click
Select multiple non-consecutive items	CTRL+click
Select multiple non-consecutive items maintaining the order of selection	SHIFT+CTRL+click
Select multiple consecutive items	SHIFT+click, SHIFT and drag, or drag
Select next optical camera	]
Select previous optical camera	[
Select next video camera	CTRL+]
Select previous video camera	CTRL+[
In a Camera view, sweep select for manual masking	ALT+drag

Shortcuts for moving the camera view

Use the following mouse actions to move the camera view in the **3D Perspective**, **3D Orthogonal**, and **Camera** views.

Task	Keys and mouse
Zoom: Move the camera viewpoint closer to or further away from the focal point	Right-click + drag forward or backward
Orbit: Move the 3D viewpoint around the focal point	Left-click + drag left, right, forward, or backward
Translate: Move the 3D viewpoint along a horizontal or vertical axis	Click wheel button + drag left, right, forward, or backward
Zoom to window (for all windows). Applies in Camera, 3D Overlay and Rotated views.	CTRL+SHIFT+Z
Zoom camera view to fit	CTRL+SHIFT+F

Shortcuts for viewing data in 3D views

Use the following hot keys to view data in the 3D Perspective and 3D Orthogonal views:

Task	Keys
Toggle display of labels	CTRL+space bar
Reset footstrike counters (see Automatically assess foot strikes on page 146).	CTRL+SHIFT+R

Shortcuts for viewing data in the Graph view

Use the following hot keys and mouse actions to view data in the Graph view. For more information, see Shortcuts for visualizing graph data on page 37.

Task	Keys and mouse
Select range of frames to zoom	ALT and right-click + drag across frames
Slide x-axis left	Click wheel button + drag left
Slide x-axis right	Click wheel button + drag right
Slide y-axis up	Click wheel button + drag forward
Slide y-axis down	Click wheel button + drag backward
Zoom x-axis in	Right-click + drag left
Zoom x-axis out	Right-click + drag right
Zoom y-axis in	Right-click + drag backward
Zoom y-axis out	Right-click + drag forward

Shortcuts for visualizing graph data

The way the graph that is displayed in a **Graph** view depends on whether the system connection is live or offline and whether an individual point or a range has been selected for plotting.

When zooming into or out of graph data, the display of grid lines in the view pane can be set to guide the eye toward the selected area of focus. Major grid lines remain at their normal weight, while any minor grid lines gradually fade. To obtain this behavior, open the **Options** dialog box (F7) and under **General View Options**, select **Graph**. In the **Properties** pane on the right, ensure **Show Minor Grid Lines** is selected.

Zoom an axis (x or y)

All component graphs in a single workspace maintain the same scale for both the x- and y-axes. The x-axis is shared across all components, but each component has its own y-axis. The y-axes may show different ranges, but represent the same number of values.

- Offline: The portion of the specific component trace displayed in the view pane is centered around the point where the mouse was clicked. All other component views are scaled by the same amount, with the vertical range centered on the median value of the visible portion of all the selected traces.
- Live: The x-axis, the workspace is centered around zero, keeping zero on the right edge of the workspace and changing the values displayed on the left.

Zoom selected range of frames

- Offline: The y-axis displays only the selected area of the specific trace and the x-axis displays only the selected frames.
- Live: This type of zooming in the x-axis is disabled to ensure that the live frame is always on the right of the graph.

Pan across an axis (x or y)

- Offline: Each component in the y-axis can be panned independently.
- Live: Panning in the x-axis is disabled to ensure that the live frame is always on the right of the graph.

Open a Quick Reports window

In addition to displaying a Graph view, you can also open a **Quick Reports** window (press F4), which enables you to display multiple graphs of model outputs normalized over the gait cycle. For more information, see Quick Reports in the *Vicon Nexus Reference Guide*.

Shortcuts for working with the time bar

Use the following hot keys and mouse actions to work with the time bar at the bottom of a view pane:

Timescale displayed in timeline

Task	Keys and mouse
Slide timeline left	Middle-click + drag left
Slide timeline right	Middle-click + drag right
Select range of frames to zoom	ALT and right-click + drag across frames
Zoom scale in	Right-click and drag right or up
Zoom scale out	Right-click and drag left or down

Time bar data displayed in view pane

Task	Keys and mouse
Start/Stop data playback	Middle-click
Jog forward/backward through data playback	Rotate mouse wheel forward /backward
Move Current Time Cursor to specific frame	Click frame in the timeline
Move Start Range Frame Cursor back to zero frame of trial	Click cursor
Move End Range Frame Cursor back to last frame of trial	Click cursor
Go to the previous frame	LEFT ARROW
Go to the next frame	RIGHT ARROW
Go to the first frame	HOME
Go to the last frame	END
Go forward 10 frames	PAGE UP
Go backward 10 frames	PAGE DOWN
Go to frame <*number*>	CTRL+G
Set Region of Interest	CTRL+D

Event identification mode in timeline

Task	Keys
Enter/exit event identification mode (where the time cursor follows the mouse)	CTRL+E
Go to the previous event	CTRL+LEFT ARROW
Go to the next event	CTRL+RIGHT ARROW
Lock/unlock event context (In event identification mode, select desired Left, Right, or General event context on timeline; subsequently moving the mouse forward or backward does not change context.)	UP ARROW or DOWN ARROW
Display context menu (after event context locked)	ENTER
Highlight command from context menu	UP ARROW or DOWN ARROW
Select highlighted command from context menu	ENTER
Move event to next frame	ALT+RIGHT ARROW
Move event to previous frame	ALT+LEFT ARROW

Shortcuts for gap-filling

Use the following hot keys to speed up gap-filling and correcting swapped marker labels:

Task	Keys
Select next gap	CTRL+8
Select previous gap	CTRL+7
Spline fill	CTRL+U
Spline fill all	CTRL+I
Pick source for pattern filling	CTRL+9
Pattern fill	CTRL+O
Pattern fill all	CTRL+P
Pick source for rigid body fill	CTRL+J
Rigid body fill	CTRL+M
Rigid body fill all	CTRL+,
Pick segment for kinematic fill	CTRL+K
Kinematic fill	CTRL+L
Kinematic fill all	CTRL+.
Cyclic fill	CTRL+;
Cyclic fill all	CTRL+'
Swap marker labels	CTRL+T

Shortcuts for using the Quality tab

Use the following hot keys to speed up working on the **Quality** tab, when using the **Data Correction** view type:

Task	Keys
Show/hide unlabeled trajectories	CTRL+F3
Show/hide trajectory names	CTRL+F4
Move to next gap for selected trajectory	CTRL+8
Move to previous gap for selected trajectory	CTRL+7
Move to previous trajectory	CTRL+PgUp
Move to next trajectory	CTRL+PgDown

Prepare a Vicon system

Before you can use your Vicon Nexus system, you need to configure the system for motion capture.

When you have done this, the next step is to prepare a hierarchy of folders in which to store all the files associated with your motion capture trials.

These procedures are explained in the following topics:

- Configure Vicon hardware in Nexus on page 44
- Prepare a data storage location on page 94

Before you begin to configure your Vicon system, ensure that the following prerequisites have been met:

- It is assumed that your Vicon system hardware (including the Vicon cameras, Vicon connectivity units, and any supported third-party devices) has been set up and connected and that Nexus has been installed and licensed. Your Vicon system may have been professionally installed by a Vicon Support engineer. If you are installing the system yourself, for full details on installing system hardware, see the Vicon documentation that was supplied with your hardware, and for installing and licensing the software, see *Installing and licensing Vicon Nexus*.
- Vicon Nexus software is licensed using Safenet licensing and the VAULT licensing system. The licensing drivers must have been installed on the host PC, and the dongle must be plugged into an appropriate port (parallel or USB) on the computer while you are running the application software.
- The IP address for the Ethernet card on the host PC must be set to 192.168.10.1 using the default IP address range (for details, see the Vicon PDF *PC Setup for Vicon Systems*, which can be downloaded from the Vicon documentation website).

Configure Vicon hardware in Nexus

The first time you use your Vicon Nexus system, you must configure the Vicon cameras, connectivity units, and any supported third-party devices such as digital video cameras, force plates, or EMG devices. After this, you only need to change the system setup in Nexus if you change your hardware configuration or if you need different system settings, for example, a different camera frequency.

You can save your system settings in a configuration file so that you can re-use or modify them later. You can create any number of system configuration files. You can then load the appropriate file for a particular type of motion capture application. For more information, see Manage configurations in Vicon Nexus on page 21.

When you start Nexus, it automatically detects all the hardware currently connected to your Vicon system, and groups them into several different categories. You configure the Vicon system hardware and system-wide parameter and data processing settings on the **System** tab in the **Resources** pane (referred to as the **System Resources** pane).

Basic setup information for each type of hardware that is likely to be present in a Vicon Nexus system is included in the following topics:

- Connect devices running the Vicon Control app on page 49
- Configure system settings on page 52
- Configure Vicon optical cameras for data capture on page 55
- Configure video cameras for digital video capture on page 61
- Aim Vicon cameras on page 64
- Mask unwanted reflections on page 66
- Configure Vicon connectivity units on page 70
- Configure supported devices on page 74

If your system contains more than one Vicon Lock+ or MX Giganet, or if you want to change the automatically assigned synchronization master, also see Change the synchronization master on page 45.

If you are not sure whether your firmware is up-to-date or if you have received an email from Vicon Support about updating your firmware, see the instructions on how to Update firmware on page 46.

Change the synchronization master

The node for the device designated as the Vicon system synchronization master, which is responsible for providing the master synchronization signal to the system, is highlighted in bold in the **System Resources** tree. Depending on the age of your Vicon system and the connectivity devices it includes, Vicon Nexus automatically designates the synchronization master:

- If a Vicon Lock+ or an MX Giganet is included in the system, it is automatically designated as the synchronization master.
- If the system does not contain a Lock+ or an MX Giganet, then a Vicon camera is automatically designated as the synchronization master.
- If the system contains more than one Lock+ or MX Giganet, you must ensure that the connectivity unit that is connected to the PC (known as the primary unit) is the synchronization master. However, because Vicon software cannot detect which connectivity unit is connected to the PC, you may need to change the automatically selected master.

To change the synchronization master:

1. At the top of the **System** tab, click **Local Vicon System** to select it.
2. At the top of the **Properties** pane, click **Show Advanced**.
3. In the **System** section of the **Properties** pane, click the **Preferred Master** list and then choose the required synchronization master from the list.

Update firmware

Each Vicon camera and connectivity unit is programmed with firmware to control its operation. Periodically, Vicon supplies firmware updates to correct or improve device functionality. You apply these firmware updates to your Vicon devices via the Vicon Ethernet network using the Vicon Firmware Update Utility, as described below.

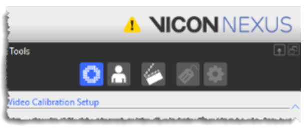
For versions of Nexus 2.8 and later, you are automatically notified when any component of your Vicon system is running out-of-date firmware, and given the opportunity to update to the latest version. (For information about updating firmware for earlier versions of Nexus, see the earlier Nexus documentation.)

> **Important**
>
> To ensure optimum performance and access to all the latest functionality, Vicon recommends that you upgrade to the latest firmware whenever it becomes available.

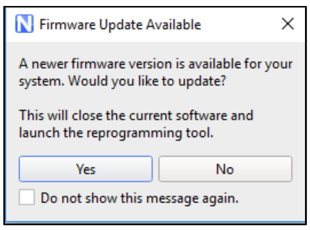
To monitor and/or upgrade system firmware:

1. When you start Nexus or connect any Vicon devices into your system, Nexus checks to see whether the firmware for all your devices is up-to-date.

   If your devices aren't using the latest firmware, Nexus displays an icon in the toolbar to let you know that a more up-to-date version of the firmware is available:

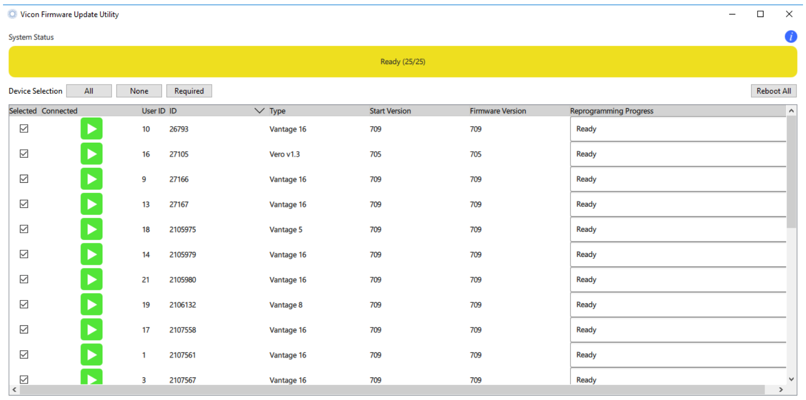
   

2. Click on the icon to display more information.
   Nexus displays a prompt that enables you to open the Vicon Firmware Update Utility (reprogramming tool).

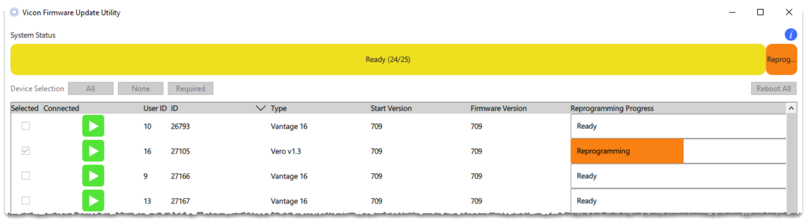
   

3. Click Yes to open the Vicon Firmware Update Utility. Note that you can also open the Vicon Firmware Update Utility from the **Start** menu (select **Vicon** > **Vicon Firmware Update Utility**).

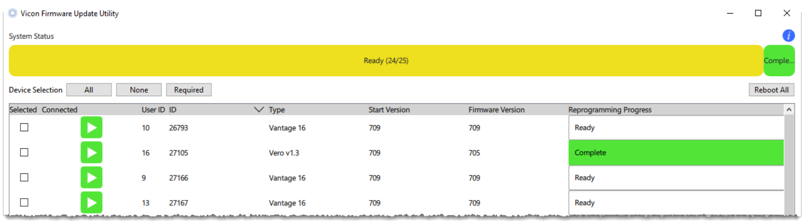
   Nexus closes and the Vicon Firmware Update Utility is displayed, showing all the connected devices and their current firmware version.
   By default, all devices are selected.



4. If you don't want to update any of the devices, clear the relevant check box(es). Note that if required, you can select devices to be updated that are already using the latest version.
5. At the bottom of the Vicon Firmware Update Utility window, in the **Choose Firmware version** list, select or browse to the required firmware version.
6. Click **Reprogram** to update the firmware for the selected device(s).



   When updating is complete, the **Firmware Version** column displays the updated firmware version and the **System Status** line and the **Reprogramming Status** column display **Complete** on a green background.



> **Note**
>
> If you do not have continual internet access, Nexus is unable to notify you when a new version of the system firmware is available. In this case, install the Vicon Firmware Update Utility on an internet-connected machine to detect and download the latest version of the firmware. You can then transfer this download to the local machine and use the Vicon Firmware Update Utility to update to the latest version of the firmware.

To downgrade to an earlier firmware version

To downgrade to a firmware version that was previously downloaded, open the Vicon Firmware Update Utility (from the **Start** menu click **Vicon** > **Vicon Firmware Update Utility**) and select the required firmware version.

Connect devices running the Vicon Control app

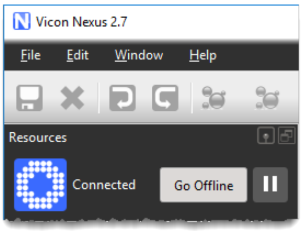
To make it easy for a single operator to set up Vicon cameras, you can use the Vicon Control app, which runs on compatible iOS and Android devices. For more information, see the Vicon Control app on the Vicon website. If you are using the Vicon Control app, connect it to your Vicon Nexus system so that you can use it to configure Vicon cameras.

The following procedure describes how to connect Vicon Control to your Vicon Nexus system.

Before you can use your phone or tablet with Nexus, you must pair it with the PC that is running Nexus (the Vicon host PC).

To connect a device running the Vicon Control app to Nexus on a Vicon host PC:

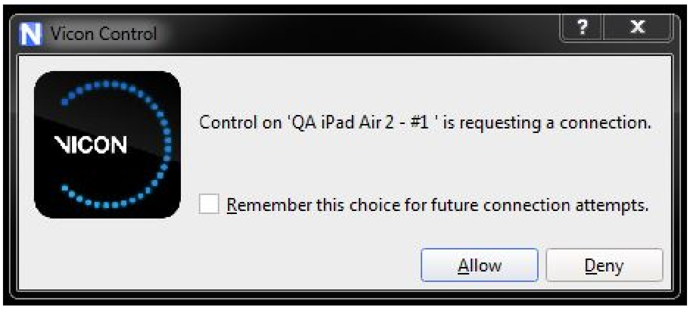
1. Ensure that your device is connected to a Wifi access point that is on the same subnet as the Vicon host PC.
2. On the Vicon host PC, ensure that the required connection is used, that Nexus is running, and the system is connected.



3. On the device, open the Vicon Control app.

The connection to Nexus is displayed on the initial Control screen:

4. Tap the Nexus icon.
   You are alerted that you must authorize the connection on the Vicon host PC before you can continue.
   In Nexus on the Vicon host PC, an authorization request is displayed:



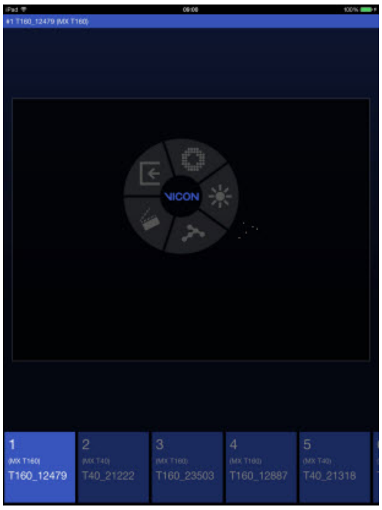
5. To use the same connection in future, select **Remember this choice for future connection attempts**. To permit Control to access Nexus, click **Allow**.

Tip

If later you need to revoke authorizations for Vicon Control, either right-click the node for the device in the **System Resources** tree and then click **Revoke Authorization**, or on the **Window** menu in Nexus, click **Manage Control Authorizations** and click **Forget Device** (for a single device) or **Forget All** (for all connected devices).

On the device, a screen similar to the following is displayed:

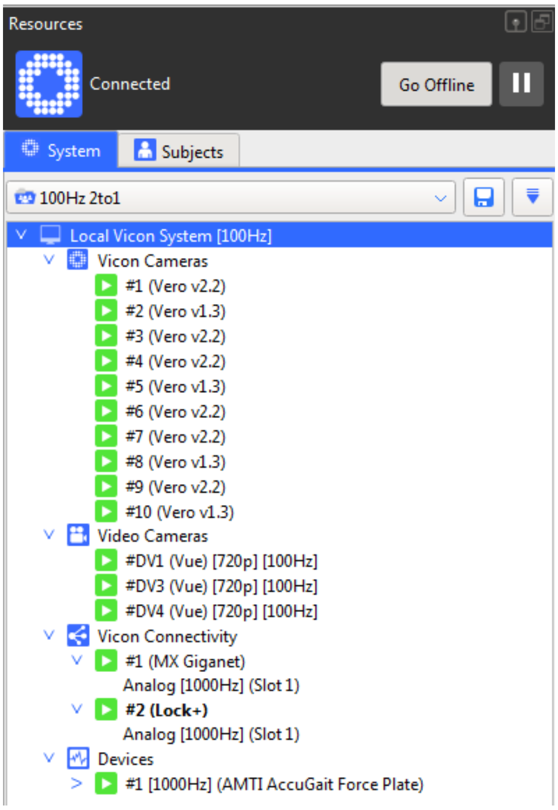
6. To select a camera and display a camera view, tap at the bottom of the screen. You can swipe the camera view right or left to change to the next or previous camera and use stretch and pinch as normal to zoom in and out.
   To access the dial control, tap and hold in a selected camera view.



7. Use the dial to view and change settings, calibrate and capture.

Configure system settings

You specify system-wide settings in the **System Resources** pane, by clicking on the top-level node called **Local Vicon System**. This node is displayed when Nexus is in Live mode (if necessary, click the **Go Live** button to see this node). It contains sub-nodes for each device connected to your Vicon system.



If you have saved any system configurations (.system files), before changing **Local Vicon System** settings, ensure the required configuration is selected at the top of the **System Resources** pane (see Manage configurations in Vicon Nexus on page 21).

The **Local Vicon System** node provides access to system-wide properties, enabling you to:

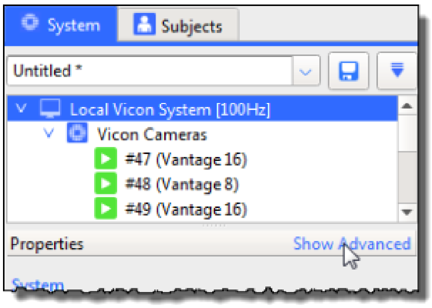
- Configure the Vicon system capture rate and the amount of memory allocated to Nexus for motion capture.
- Manage the way Nexus produces real-time 3D representations of the subjects whose motion is being captured.
- Specify the identification and connection settings for the Nexus host PC.
- Specify video standards and timecode options.

To change Vicon system settings:

1. If Nexus is currently offline, in the **Resources** pane, click **Go Live**.
2. On the **System** tab, click the **Local Vicon System** node.
3. In the **Properties** pane at the bottom of the **System** tab, view or change settings for the required properties to suit the needs of your motion capture application. When you set up your Vicon system, ensure that the **Requested Frame Rate** property in the **System** section of the **Properties** pane is suitable for your application.
   The **Requested Frame Rate** is the rate (in Hz) at which to synchronize the Vicon cameras. If you are using an external video signal, select from displayed values (multiples of the base frame rate of the PAL, NTSC, or Film video standard specified in Standard) up to a maximum of 2,000. You can choose any number you want if you do not have any Genlock Standard set.

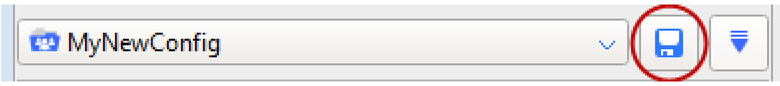
   > **Tip**
   >
   > The default setting of 100 (Hz) is suitable for a range of common applications, so unless you have a specific requirement for a different value, you do not need to adjust this setting.

   You can configure additional properties as required to suit your motion capture application. To view all the available properties for **Local Vicon System**, at the top right of the **Properties** pane, click **Show Advanced**.

   For detailed information about each of the properties, see *Local Vicon System properties* in the *Vicon Nexus Reference Guide*.

When you have finished specifying the properties, at the top of the **System** tab, click the **Save** button to save your system configuration settings to a *.system* file in the Systems configurations folder (see Manage configurations in Vicon Nexus on page 21).

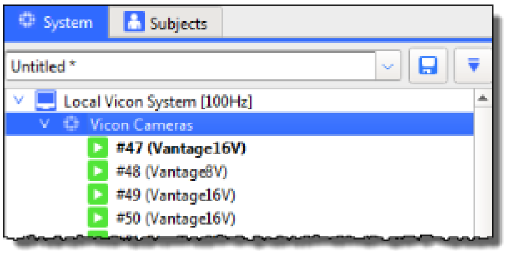


Tip

The node for the device designated as the Vicon system synchronization master is highlighted in bold in the **System Resources** tree. For information on changing the synchronization master, see Change the synchronization master on page 45.

Configure Vicon optical cameras for data capture

As part of setting up your Vicon system, you must specify the required settings for Vicon optical cameras. To do this, you use the **Vicon Cameras** node in the **System Resources** pane. You can configure the settings for an individual camera, several cameras, or all cameras at once.



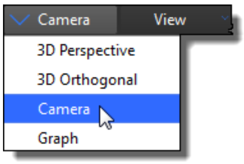
The **Vicon Cameras** node is displayed under the **Local Vicon System** node when Vicon Nexus is connected to a Vicon system and is in Live mode. It is displayed under the **Vicon Data** node when Nexus is in Offline mode. It lists each Vicon optical camera connected to your Vicon system. For each camera, the node name includes the device position number, any display name specified in the **Identification** property, and the camera type in parentheses, for example, **#1 Over Door (Vantage 16)**.

> **Important**
>
> Before making changes to your Vicon camera settings, ensure that:
>
> - You have set your cameras' focus and aperture. (To see how to set up Vicon Vero cameras with Vicon Tracker software, which is similar to Nexus setup, see the Tracker Installation and Training Guide on YouTube.)
> - The required system configuration has been selected in the **System Resources pane** (see Manage configurations in Vicon Nexus on page 21).
> - Your calibration device (wand) is available.
> - You have some markers with which to outline the capture volume. This will also assist with camera setup.

If you are setting up a mixed camera system (that is, a system that includes both Vicon MX T-Series cameras as well as other current Vicon cameras), see also Set up mixed Vicon camera systems on page 59.

To configure Vicon optical cameras for data capture:

1. Ensure Vicon Nexus is in Live mode. If it is not, in the **Resources** pane, click **Go Live**.
2. To visualize your capture volume, from the view pane menu, select **Camera**.



3. In the **System Resources** tree, select the node(s) for the camera(s) whose properties you wish to configure, either:
   - **Vicon Cameras** node for all Vicon cameras

   or

   - A sub node for a specific Vicon camera. The camera sub-nodes on the **System** tab correspond to the **Names** you set in the **Properties** pane (see below). Note that If no Lock+ or MX Giganet is present in the Vicon system, the sub-node for the Vicon camera acting as the synchronization master is displayed in bold.

   When a camera is selected, a blue status light on its strobe unit lights up.

   > **Tip**
   >
   > In many cases, it is best to start by selecting all of the cameras, to find a common baseline. You can then adjust individual cameras as required.

4. In the capture volume, have someone wave the calibration wand and ensure that you can see marker images moving in the **Camera** view.

5. In the **Properties** pane at the bottom of the **System** tab, click **Show Advanced** to show additional properties.
   When you first set up your Vicon system, configure the following camera properties in the order shown. (If you are not sure what a particular setting means, you can display a tooltip by hovering the mouse over the relevant field or control):
   Identification section:

   - **Name** (If you wish to distinguish it from the other cameras)

   Settings section:

   - **Strobe Intensity** In most cases, keep its default setting (1). However, if your Vicon system consists of a mix of MX T-Series cameras and other current Vicon cameras, and if it is crucial to your work that the shutter periods for all are precisely aligned, ensure that your firmware is upgraded to version 700 or later, and set the **Strobe Intensity** for the T-Series camera(s) to the maximum. For more information, see Set up mixed Vicon camera systems on page 59.
   - **Gain** Only adjust this setting if the markers appear too faint or the cameras have trouble distinguishing them; otherwise, leave at its default setting (x1). Vicon does not recommend using a setting higher than x2.
   - **Grayscale Mode** This setting determines what data is sent from the camera to the computer. Ensure this is set to the default setting (**Auto**) for capturing data. If the camera recognizes a blob as a circle, only centroid data is sent. If the camera cannot distinguish the blob as a circle, full grayscale data is sent so that Nexus can attempt to circle-fit the blobs. Circle fitted markers are displayed as crosshairs in the **Camera** view.

   Centroid Fitting section:

   - **Threshold** This setting differentiates between markers and ambient light. A value in the region of 0.2 (the default) to 0.5 is usually appropriate, but Vicon strongly recommends that you view static markers in the volume to establish an appropriate setting. If cameras are evenly spaced around the volume, the same **Threshold** value is usually sufficient for all cameras.
   - **Minimum Circularity Ratio** The circularity threshold used by the centroid-fitting algorithms in a Vicon camera to fit centroids to grayscale blobs. The higher the value, the more stringent the centroid fitter is. For camera calibration, you may wish to apply higher settings to ensure that the Vicon system selects the best markers and thus provides the best possible calibration. For data capture, a lower value may be appropriate. When a blob is fitted with a centroid, it is represented by crosshairs. The default setting is 0.5.

> **Tip**
>
> If adjusting these settings does not easily enable you to eliminate reflections, create camera masks to eliminate reflections and other unwanted light sources that occur in parts of the capture volume. For information on masking, see Mask unwanted reflections on page 66.

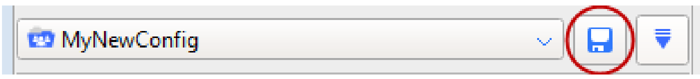
Centroid Tracking section:

- **Enable Centroid Tracking** Tracking 2D camera centroids provides extra information that maintains marker labels in real time when only one camera can see a marker. When enabled, the 2D track calculations are performed by a camera's onboard sensors. When disabled, the 2D track calculation is performed by the PC (in Nexus). The default is off.
- **Marker Velocity** Maximum velocity at which a marker will be tracked, expressed as the percentage of image width per second. The default is 5.

> **Important**
>
> These properties affect the quality of the motion capture data. You cannot adjust them after data capture, so it is important to optimize these before you collect data intended for analysis. In subsequent sessions, you may wish to configure additional properties to suit the needs of your motion capture application.

For further details about each of the **Vicon Camera** properties, see *Vicon Camera properties* in the *VIcon Nexus Reference Guide*.

6. When you have finished adjusting the **Vicon Cameras** settings, in the **Settings** section, ensure that **Grayscale Mode** is set to **Auto**.
7. At the top of the **System** tab, click the Save button to save your system configuration settings to a .system file in the Systems Configurations folder (see Manage configurations in Vicon Nexus on page 21).



Set up mixed Vicon camera systems

Vicon Nexus (version 2.4 and later) enables you to run mixed Vicon camera systems consisting of Vicon Vero cameras (v1.3 and v2.2), Vicon Vantage cameras (V5, V8, V16) and/or MX T-Series cameras (T10, T20, T40, T160, or S Edition) and Bonita Optical cameras (B3, B10). You can also use Vicon Vue and Bonita Video cameras in the same mixed system.

> **Caution**
>
> The use of mixed systems that include Vicon cameras older than T-Series and Bonitas is not supported and full functionality cannot be guaranteed.

For systems involving only Vero, Vantage and Bonita cameras, the shutter period characteristics for all cameras match exactly. Irrespective of individual cameras' strobe (shutter) settings, the center alignment of these periods in any Vero/Vantage/Bonita camera in the same system align exactly. You do not need to make any adjustments to ensure that this alignment occurs.

However, for systems involving Vicon MX T-Series cameras, depending on your requirements (see When are differences in strobe timings important? on page 60), you may need to make some manual adjustment (see the following steps).

> **Important**
>
> Support for mixed systems' center strobe alignment requires Vicon firmware 700 or later. Vicon recommends that you always update to the latest firmware.

Due to the differences in strobe timings between the current Vicon cameras (Vero, Vantage and Bonita cameras) and the MX T-Series cameras, in situations where very small timing differences are considered to be relevant and greater than other accepted limitations (such as skin movement artifacts), ensure that the camera strobe periods match by setting the **Strobe Intensity** for the MX T-Series camera(s) to maximum as described below.

To obtain consistent strobe timing and sensor exposure in mixed camera systems that include T-series:

1. In the **System Resources** tree, select the MX T-Series camera(s).
2. In the selected camera's **Properties** pane, in the **Settings** section, ensure the **Strobe Intensity** is set to its maximum.

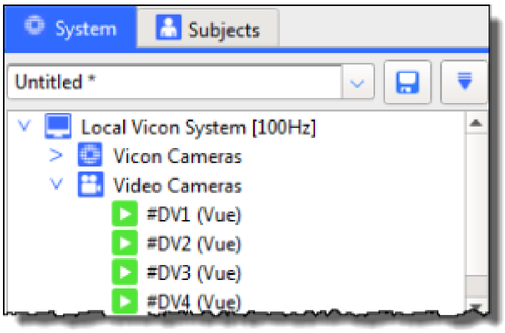
This ensures that the center of the strobe pulse and shutter period for the Vero/Vue /Bonita cameras matches that of the MX T-Series cameras.

When are differences in strobe timings important?

In situations where very small timing differences are considered to be relevant and greater than other accepted limitations (such as skin movement artifacts), ensure that all the camera strobe periods match by setting an appropriate value in the **Strobe Intensity** for the cameras you are using. If all cameras are of the same type, this value is the same for all cameras, but for systems that include both T-series and other current Vicon cameras, set the TSeries' **Strobe Intensity** to its maximum, as described above. Situations that may warrant this consideration include studies where very fast ballistic movements are expected and/or where very small markers are likely to be in close proximity.

Configure video cameras for digital video capture

You manage the identification and configuration settings for each digital video camera connected to your Vicon system with the **Video Cameras** node in the **System Resources** pane. Using this node, you can also remove or reorder video cameras.



> **Note**
>
> References to "Vicon video camera" apply to the use of Vicon video cameras (Vicon Bonita and Vicon Vue). The following instructions also apply to Basler cameras that are supported by Vicon Nexus. For details of how to set up a Vicon system that includes video cameras, see the PDF PC Setup for Vicon systems.

The **Video Cameras** node is displayed under the **Local Vicon System** node when one or more video cameras is connected to the Nexus host PC and Vicon Nexus is in Live mode, or when a previously saved file containing video cameras is loaded. It lists each video camera connected to your Vicon system. For each camera, the node name includes the camera number, the camera type in parentheses and the frame rate in brackets, for example, **#DV2 Over Door (Bonita 720c) [50Hz]**.

> **Important**
>
> For this node to be displayed, Nexus must be in Live mode (click the **Go Live** button) or a previously saved trial must be loaded. Before managing video cameras in your Vicon system, ensure that the desired system configuration has been selected in the **System Resources** pane (see Manage configurations in Vicon Nexus on page 21). This topic assumes that the digital video camera is connected and feeding data through the FireWire or GigE port. For information on setting up this port, see the documentation provided with your FireWire card or GigE port.

You can connect a digital video camera to the Nexus host PC. You can connect supported Basler video cameras to the FireWire or GigE port on the PC and then capture reference video for use with your optical motion capture data. You can overlay the Vicon optical motion data onto the digital video images from the video cameras in the **Camera** view using the **3D Overlay** option, from the **View** menu. If required, you can burn this information into the digital video image and export the .avi file for viewing in another application using the **Configure the Export 3D Overlay Video** operation.

To configure video cameras for digital video capture:

1. Ensure Nexus is in Live mode (in the **Resources** pane, click **Go Live**).
2. In the **System Resources** tree, select the node whose properties you wish to configure:
   - **Video Cameras** node, for all digital video cameras
   - A sub-node, for a specific video camera

   Note that if you are configuring a Vicon video camera, the node will not become green until you specify the **Destination IP Address**.
   Also note that cameras with gray icons are connected to the network but are not contributing data. For example, if you have connected Vicon Bonita video cameras, but your Vicon connectivity device does not support jumbo packets, the icons for the Vicon video cameras will remain gray. For more information and help with upgrading your system, contact Vicon Support.
3. In the **Properties** pane at the bottom of the **System Resources** pane, view or change settings for the required properties.
   When you first set up your Vicon system, you must configure at least the following properties in the order shown. In subsequent sessions, you may want to configure additional properties to suit the needs of your motion capture application.

Section	Property name
Hardware	**Destination IP Address** (Vicon video cameras)
Settings	▌**Capture Path** (enter a separate HDD or an SSD drive for each camera) ▌**Video Gain** and **Brightness Offset** (Vicon video) ▌**Camera Gain** and **Camera Brightness** (Basler)
Frame Rate:	**Requested Frame Rate**
Settings	**Shutter Duration** and **Camera AOI** (Basler),
Frame Rate	**Packet Size** (Baslers using Firewire)
Hardware	**Trigger Source** (Basler)
Calibration	**Focal Length** (millimeters) (if you will be using **Aim Cameras**)

For further details about each of the **Video Camera** properties, see *Video Camera properties* in the *Vicon Nexus Reference Guide*.

4. If you are using Basler cameras, to align the shutters with the rest of the Vicon system, in the **System Resources** tree, right-click the **Video Cameras** node and then click **Align Shutters (Basler)**.
5. At the top of the **System** tab, click the Save button to save your system configuration settings to a *.system* file in the **Systems** configurations folder (see Manage configurations in Vicon Nexus on page 21).

> **Tip**
>
> When you plug a video camera into the Vicon system, the system automatically assigns it a number. To reorder video cameras, select the camera in the **Reorder Devices** dialog box and click **Move Up** or **Move Down**. Then click OK.

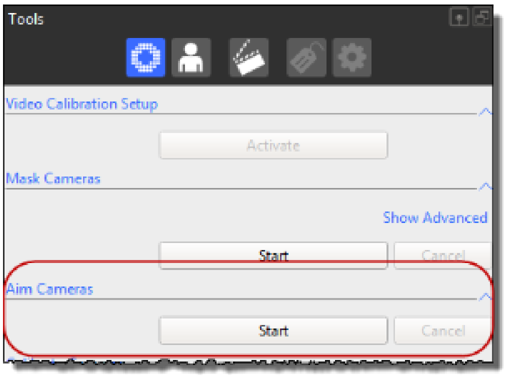
Aim Vicon cameras

> **Note**
>
> Aiming cameras is useful for providing an initial, approximate calibration, before you fully calibrate the cameras as described in Calibrate Vicon cameras on page 97.

Before you aim your Vicon cameras, ensure that they are displayed in the **3D Perspective** view.

Vicon camera positioning changes in real time. When you physically move a camera in the capture volume, you should be able to see that its representation moves correspondingly in the **3D Perspective** view. If you can't see the cameras in the **3D Perspective** view, or if you want to change the way the cameras are displayed, press F7 to open the **Options** dialog box, and on the left, ensure **Camera Positions** is selected. If required, change the **Properties** on the right.

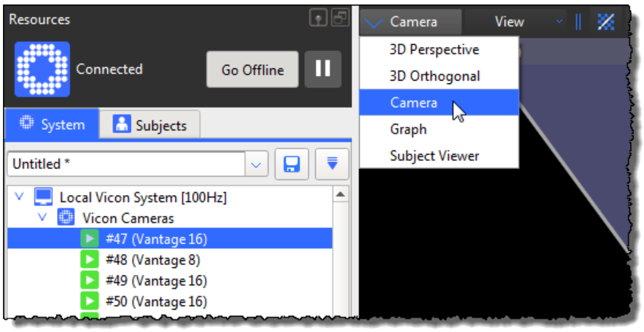
When you have roughly positioned the cameras in the volume, you can use the **Aim Cameras** section of the **System Preparation Tools** pane to optimize their positions, before running a full camera calibration.



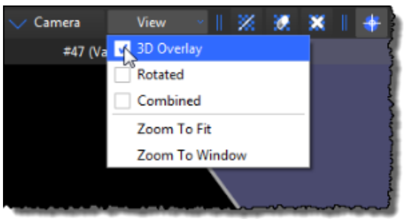
> **Important**
>
> To aim Vicon cameras, you must use the calibration device supplied with your Vicon system. To aim digital video cameras, you must use an Active Wand.

To optimize the position of Vicon cameras:

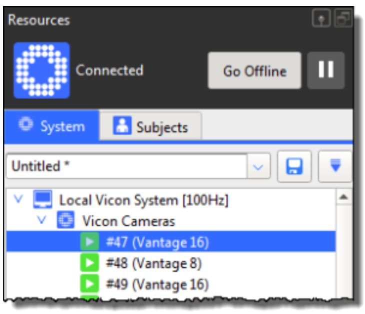
1. In the **Resources** pane, ensure Nexus is in Live mode.
2. In the **Options** dialog box (press F7), under the **General View Options** section, select the **Target Volume** option.
3. On the right side of the dialog box, update the properties of the target volume to reflect the dimensions of your capture volume.
4. View the Vicon camera feedback by doing the following:
   a. In the view pane, select the **Camera** view.



   b. In the **Camera** view toolbar, on the **View** drop-down list, select **3D Overlay**.



   c. In the **System Resources** tree, click the cameras that you want to aim to select them.



   A virtual representation of your target volume is overlaid on the 2D data from the camera image.

5. In the **System Preparation Tools** pane, expand the **Set Volume Origin** section and from the **L-Frame** drop-down list select the **Active Wand** or **5 Marker Wand & L-Frame** calibration object.
6. In the capture volume, place the calibration object flat on the floor in the center of the capture volume at the desired origin position.
   In the **Aim Cameras** section, click **Start** to begin the camera-aiming process. Nexus starts attempting to identify the calibration object in each camera view, and the **Start** button switches to its **Stop** setting.
7. In the **System Resources** tree, select the camera you want to position and ensure that you can see the calibration object in the **Camera** view.
8. Physically move a Vicon camera in the capture volume and check its coverage against the target volume.
9. Repeat steps 7–8 for each Vicon camera, until they are all correctly aimed at the calibration object.

Mask unwanted reflections

Masking enables grayscale blobs that are generated from reflections of objects other than markers (eg, reflections from floor or furniture, or opposing camera strobe rings) to be ignored by the Vicon cameras. In capture volumes where this spurious data is present, camera masking improves calibration robustness and reconstruction quality.

In Nexus, masking is shown as a grid of small blue tiles superimposed over the camera image in a **Camera** view, each cell of which can be set to obscure unwanted reflections that are seen by a camera.

Vicon Nexus offers both automatic masking and tools for manual masking. If you have a large number of reflections in your capture volume, it is a good idea to create camera masks automatically first. You can then eliminate any remaining reflections by creating masks manually.

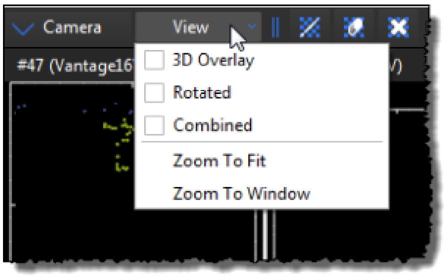
For more information, see:

- Automatically create Vicon camera masks on page 67
- Manually create Vicon camera masks on page 68

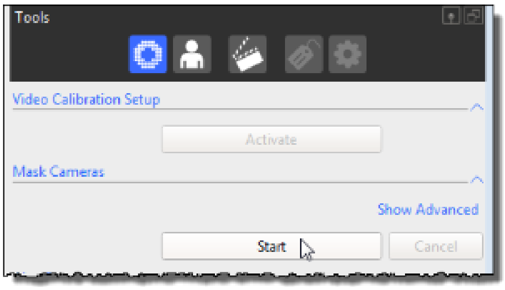
Automatically create Vicon camera masks

To automatically create camera masks:

1. Remove from the capture volume any unnecessary objects, such as calibration objects. For best results, the capture volume should be entirely free from objects likely to cause background interference.
2. In the **Resources** pane, ensure Nexus is in Live mode.
3. From the **System Resources** tree, select all Vicon cameras.
4. From the view pane toolbar, select **Camera** to display the 2D data being captured by each selected camera in a separate **Camera** view.
5. From the **View** drop-down list at the top of the view pane, ensure that the **3D Overlay** and **Combined** options are cleared.



6. In the **Options** dialog box (F7), under the **General View Options** section ensure that the **Threshold Map** option is selected.
7. Ensure you have adjusted camera settings as described in Configure Vicon optical cameras for data capture on page 55. Any reflections are clearly visible in the **Camera** view, typically as non-circular areas of grayscale or edge data. Note that as reflections can severely affect the camera data rates, you may find that the camera overloads. In this case, the camera automatically sends edge data instead of full grayscale.
8. In the **System Preparation Tools** pane, in the **Mask Cameras** section, click **Start**.



9. The **Start** button switches to its **Stop** setting. Nexus starts recording the data visible to each of the cameras connected. Any camera masks created are displayed as blue cells in the **Camera** views for affected cameras. If there is no data visible to a particular camera, Nexus does not create any masks for it. About 30 seconds of recording is generally sufficient to enable Nexus to collect the data visible to the cameras.
10. After about 30 seconds, click **Stop**. If you still see unwanted reflections, you can start and stop the **Mask Cameras** option again until you have hidden the reflections, or mask any remaining reflections manually.

Manually create Vicon camera masks

You can manually create masks to eliminate any reflections in the capture volume that are visible to the cameras. If you have a large number of reflections in your capture volume, it is a good idea to create camera masks automatically first (see above).

To manually create camera masks:

1. From the capture volume, remove any unnecessary objects, such as calibration objects. For best results, ensure the capture volume is entirely free from objects likely to cause unwanted reflections.
2. From the **System Resources** tree, select all Vicon cameras.
3. From the view pane toolbar, select **Camera** to display the 2D data being captured by each selected camera in a separate **Camera** view.
4. From the **View** drop-down list at the top of the view pane toolbar, ensure that the **3D Overlay** and **Combined** options are cleared.
5. In the **Options** dialog box (F7), under **General View Options**, ensure that **Threshold Map** is selected.

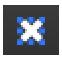
> **Tip**
>
> The **Threshold Map** default color is blue, but you can change the color in the **Options** dialog box. In the following descriptions, the masking tiles are described as blue.

6. Ensure you have adjusted camera settings as described in Configure Vicon optical cameras for data capture on page 55. If reflections are present, they should be clearly visible in the **Camera** view, typically as non-circular areas of grayscale or edge data. Note that as reflections can severely affect the camera data rates, you may find that the camera overloads. In this case, the camera automatically sends edge data instead of full grayscale.

7. From the view pane toolbar, use the following buttons to obscure any unwanted reflections visible to the selected Vicon cameras. (When you click any of these buttons, a grid of small blue tiles is superimposed over the camera image in each **Camera** view.)

   - Paint a mask onto the camera 
     Click an individual tile, click and drag across multiple tiles, or hold down ALT and click while dragging the mouse across an entire area of unwanted reflections visible in the camera grid. You can drag the mouse horizontally, vertically, or diagonally.
     When a cell is painted, its background color changes from black to blue. The camera mask consists of blue cells obscuring unwanted reflections.

   - Erase a mask from the camera 
     Click an individual tile, click and drag across multiple tiles, or hold down ALT and click while dragging the mouse across an entire area of blue cells in the camera grid. You can drag the mouse horizontally, vertically, or diagonally.
     When an individual cell is erased, its background color changes from blue to black, and any reflection that had previously been obscured is visible again.

   - Clear the mask from the camera 
     Click the button to automatically remove the entire mask from the camera.
     When the mask is cleared, the background color of any previously painted cells changes from blue to black, and any reflections that had previously been obscured are visible again.

   > **Tip**
   >
   > To zoom in on the images, right-click and drag the mouse forward (to zoom in) or backward (to zoom out). To pan the image, click both mouse buttons and drag

   The camera masks are applied in real time and are saved along with your camera calibration.

Configure Vicon connectivity units

Vicon connectivity units are smart boxes that can be combined to create a distributed architecture, enabling you to customize the number of Vicon cameras and supported third-party devices in your Vicon system.

As part of setting up your Vicon system, you configure your Vicon connectivity unit(s) for system communications and any GPO or remote triggering.

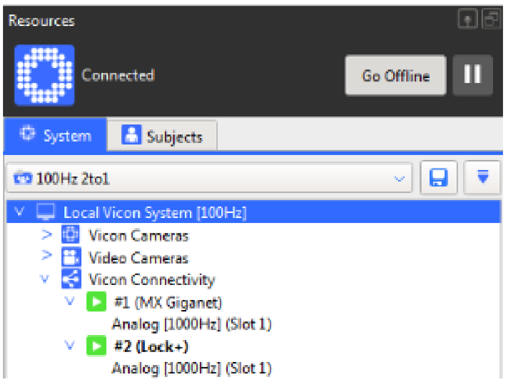
The **Vicon Connectivity** node is displayed under the Local Vicon System node when Vicon Nexus is connected to the Vicon system and is in Live mode. It is displayed under the Vicon Data node when Nexus is in Offline mode. It lists each connectivity unit connected to your Vicon system.

Depending on the type of Vicon system you are running, your Vicon system architecture will contain one or more of the following Vicon connectivity units:

- **Vicon Lock unit**: Facilitates the integration of synchronous third-party equipment with Vicon Vantage, Vicon Vero, and Vicon Bonita cameras, by providing or receiving synchronization and/or timecode. Also provides connectivity for third-party analog capture sources, such as force plates and EMG equipment. Connects to a PoE+ switch to which Vicon cameras and the host PC are connected.
  If your Vicon system includes PoE or PoE+ switches, note that although they do not appear as connectivity nodes in the **System Resources** tree, the cameras that are connected to them are displayed.

  > **Note**
  > Except where noted, references to Vicon Lock, Lock units, and Lock apply to all current models of the Vicon Lock unit (at the time of publication, this includes Vicon Lock+, Vicon Lock Studio and Vicon Lock Lab).

- **MX Giganet**: The primary connectivity unit in an MX T-Series system. To connect and synchronize force plates and EMG devices, MX Giganets can be configured with a 64-channel analog card. (This functionality is integral to Vicon Lock+ and Lock Lab units.) You can incorporate units and components from an MX T-Series system into a Vicon Vantage system. See the *Vicon Vantage Reference Guide* or contact Vicon Support for details on configuring a combined architecture.



If your Vicon system includes PoE or PoE+ switches, note that although they do not appear as connectivity nodes in the System Resources tree, the cameras that are connected to them are displayed.

Important

Before managing Vicon connectivity units, ensure that the desired system configuration has been selected in the **System Resources** pane (see Manage configurations in Vicon Nexus on page 21).

Set up Vicon connectivity units

The **Vicon Connectivity** node lists each MX Giganet or Vicon Lock unit connected to your Vicon system. For each unit, the node name includes the device position number, any display name specified in the Identification property, and the device type listed in parentheses, for example **#1 Name (Lock+)**. For Lock Lab and Lock+, and for MX Giganet if either or both analog option cards are installed, the sample rates are displayed in brackets, and an Analog Card (Slot 1) sub node is displayed as appropriate. If no analog source is selected, [No Source] is displayed after the device name.

The Lock or MX Giganet node enables you to manage the identification and configuration settings for each connectivity unit included in your Vicon system architecture. The node is displayed under the **Vicon Connectivity** node in the **System Resources** pane when Vicon Nexus is connected to a Vicon system with at least one Lock unit or MX Giganet unit and is in Live mode (click the **Go Live** button).

To configure connectivity units for analog data acquisition:

1. In the **System Resources** pane, click the **Go Live** button.
2. In the **System Resources** tree, select the node whose properties you want to configure:
   - **Vicon Connectivity** node for all Vicon connectivity units
   - A sub node for a specific connectivity unit – the sub nodes in the **System Resources** tree correspond to the IDs assigned by Nexus. If a connectivity unit has automatically been designated as the synchronization master for the Vicon system, its node name is displayed in bold.
     The colored icon beside a connectivity node identifies the status of the device:
     - Green play button: Component OK (active). If an analog device is connected, this status does not reflect the analog device's status.
     - Yellow pause button: Component is not fully set up or device has been disabled in Status section of Properties.
     - Gray play button: Component connected but not contributing any data.
     - Red stop button: Component down (unavailable or disconnected).

   In the **Properties** section at the bottom of the **System Resources** pane, view or change settings for the desired properties.

When you first set up your Vicon system, you must configure at least:

- **Name** A name you supply, to enable you to identify this unit easily
- **Sync Out** (if you are using synchronization functionality) The general purpose output driver (*.*gpo* file) you specify here determines the output frequency of the synchronization pulse. You can select a driver for each of the sync outputs. For further details, see *Go Further with Vicon MX T-Series* or *Vicon Vantage Reference Guide*.)

In subsequent sessions, you may wish to configure additional properties to suit the needs of your motion capture application. For details about all the properties for Vicon connectivity units, see Vicon Lock and MX Giganet properties in the *Vicon Nexus Reference Guide*.

3. In the **System Resources** pane, click the **Save** button to save your system configuration settings to a .*system* file in the Systems configurations folder (see Manage configurations in Vicon Nexus on page 21).

Configure supported devices

As part of setting up your Vicon system, you configure supported devices using the **Devices** node in the **System Resources** pane.

The **Devices** node is displayed under the **Local Vicon System** node when Vicon Nexus is connected to a Vicon system with at least one Vicon or supported third-party analog or EMG device and is in Live mode. It is displayed under the **Vicon Data** node when Nexus is in Offline mode. The **Devices** node lists each supported device connected to your Vicon system.

For each device, the node name includes any display name specified in the **Name** property, its approximate sample rate in brackets, and the device type in parenthesis, for example, **Name [2000Hz] (Analog EMG)**. The default name property displayed is **Name [1000Hz] (Analog EMG)**.

For analog devices, if no analog source is selected, **[No Source]** is displayed after the device type. **Channel** sub nodes are displayed for any analog channels. Predefined configurations for some devices are supplied with Vicon Nexus.

> **Tip**
>
> To add new device entries to this node, Vicon Nexus must be in Live mode (click the **Go Live** button). Before managing other devices, ensure that the desired system configuration has been selected in the **System Resources** pane (see Manage configurations in Vicon Nexus on page 21).

To view a graph of one or more device output or component signals:

1. In the **System Resources** tree, expand the **Devices** node to show the devices connected to your Nexus system
2. Select one or more devices, outputs or components.
3. From the view pane toolbar, select **Graph**.
4. If necessary, in the **Graph** view toolbar, from the **Graph Type** drop-down list, select **Components**.
5. The **Graph** view displays a graph of the selected components.

For more information on configuring supported devices, see:

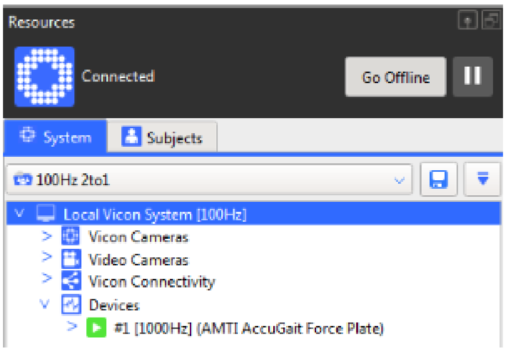
- Configure force plates on page 75
- Configure EMG devices on page 87

- Configure IMUs on page 89

Configure force plates

As part of setting up your Vicon system, you manage the connection and configuration settings for supported force plates in your Vicon system with the appropriate force plate node. You can select the required force plate node under the Devices node in the **System Resources** pane when Vicon Nexus is connected to a Vicon system with at least one force plate unit and is in Live mode. When Nexus is in Offline mode, the force plate node is displayed under the **Devices** node of the **Vicon Data** node.

The **Devices** node lists each force plate connected to your Vicon system. For each device, the node name includes any display name specified in the Name property, its sample rate in brackets, and the force plate type in parentheses, for example, **Name [1000Hz] (AMTI AccuGait Force Plate)**. If no analog source is selected, **[No Source]** is displayed after the device type. **Channel - #** sub nodes are displayed for each channel. Predefined configurations for some supported force plates are supplied with Vicon Nexus.



Important

To add new force plate entries to the **Devices** node, Nexus must be in Live mode (click the Go Live button). Before managing force plates, ensure that the desired system configuration has been selected in the **System Resources** pane (see Manage configurations in Vicon Nexus on page 21).

Force plates data can be acquired through the analog capture functionality of a Vicon connectivity device (Vicon Lock+ or MX Giganet). This topic describes the analog connection only. If your force plate has a digital output, it is possible that this data stream can also be captured, but this depends on the data stream's format and the equipment's manufacturer. For information on the digital data streams that can be used with Vicon systems, contact Vicon Support.

To configure force plates for analog data capture:

1. In the **System Resources** pane, click the **Go Live** button.
   The first time you use Nexus, the **Devices** node is empty. You must add and configure a new force plate device before it will be displayed in the **System Resources** tree.
2. In the **System Resources** tree, right-click the **Devices** node, point to **Add Analog Device** and from the context menu select the type of force plate that is integrated in your Vicon system.
   The selected force plate node automatically expands to display the newly created device. (For Motekforce Link treadmills, two new device nodes are displayed, one for each plate.) If the appropriate type is not displayed, contact Vicon Support.
   The colored icon beside a force plate node identifies the status of the device and of any connected analog source:
   - Green play button: Component OK (active or connected); if an analog device is connected, the analog source is selected and all channels are configured.
   - Yellow pause button: Component is not fully set up (e.g., not all channels have been assigned a pin or the calibration matrix has not been configured).
   - Gray play button: Component connected but not contributing any data.
   - Red stop button: Component down (unavailable or disconnected).

   These colored icons correspond to those used for the device summary in the **Status Communications** pane.
3. In the **Properties** section at the bottom of the **System Resources** pane, view or change settings for the following properties:
   a. In the **General** section, enter a Name.
   b. In the **General** section, go to the **Calibration File** field, and load the manufacturer's calibration file:
      - If the calibration file is not listed in the drop-down list, click the ellipsis (...) next to the **Calibration File** field to browse to the relevant location, and select it. The file becomes available in the drop-down list.
      - Select the calibration file. (For Motekforce Link treadmills, note that a single calibration file is used for both plates.)
      - If not supplied by the manufacturer's calibration file, you can set the device **Dimensions**, **Position**, **Orientation**, and **Origin** in those **Properties** sections.

c. If there is no calibration file, click Show Advanced at the top right of the Properties section and enter the Calibration Matrix 6x6 Matrix values manually.

> **Important**
>
> The Matrix values must be entered, either via a calibration file or by manual entry, in order for the force plate to become active.

d. In the Source section, select a Source (the Vicon connectivity device to which the device is attached) from the drop-down list.

   The Source drop-down list contains all connected Vicon connectivity devices; a USB force plate will have its Source drop-down list populated with connected USB devices of the required type.

e. In the Source section, use the Fill button to populate the input connections sequentially (if these are consecutive on the Vicon connectivity device).

f. In the Source section, select the Gain for the Source from the choice of gains available for the Vicon connectivity device.

4. In the System Resources tree, if necessary, expand the force plate node to expose the Force, Moment and CoP (Center of Pressure) channels.
   The colored icon beside the output node identifies the analog channel status:
   - Green arrow: Analog channel connected to source device.
   - Yellow arrow: Analog channel has not been assigned a pin.
5. To tare the force plate at zero load, in the General section, click the Zero Level browse button and enter the matrix properties.

> **Tip**
>
> You can also tare the force plate by right-clicking on the force plate name in the System Resources tree and selecting Zero Level.

6. In a 3D Perspective view, ensure that a gray or colored rectangle with the number 1 on it, representing the force plate, is displayed in the capture volume where you have positioned it.
   If you have multiple force plates configured, they will be numbered in the order they appear in the System Resources tree.

7. In the capture volume, have someone step onto the force plate. You should see the force vector being displayed in real time.
8. At the top of the **System Resources** pane, click the Save button to save your system configuration settings to a *.system* file in the **Systems** configurations folder (see Manage configurations in Vicon Nexus on page 21).
9. From the **System Resources** tree, expand the force plate node and select the **Force** output.
10. Switch to a **Graph** view.
11. If necessary, select **Components** from the **Graph Type** drop-down list. A real-time graph of the **Force** output is displayed.
12. Verify that the vertical (Fz) force component is equal to [known mass * 9.81].

For more information, see:

- Force plate minimum setup requirements on page 79
- Change force plate display options on page 80
- Visualize and record Force Threshold on page 83
- View combined output from multiple force plates on page 84
- Usage recommendations for Motekforce Link treadmill on page 86

See also Force plate properties in the *Vicon Nexus Reference Guide*.

Force plate minimum setup requirements

For a force plate to appear in a **3D Perspective** view, you must configure at least the following properties:

- Source
- Calibration Matrix
- Position (X, Y, Z)
- Name: Assigning a force plate name is recommended, especially if you are setting up more than one force plate.

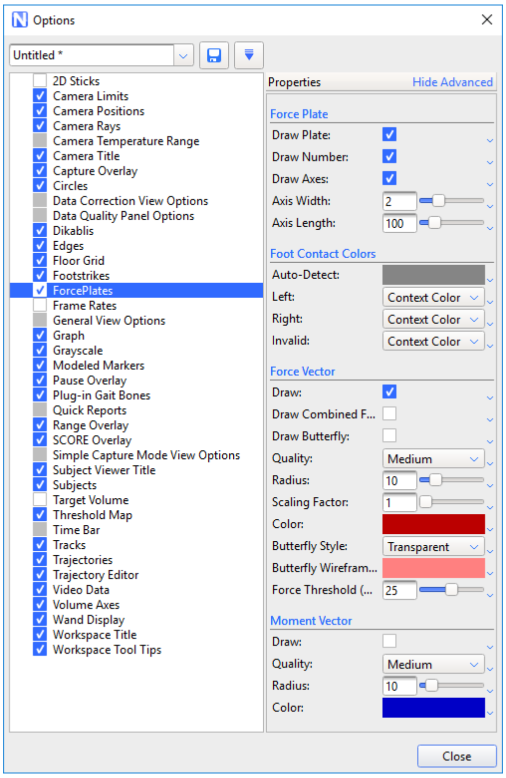
For a force plate to appear in Live mode in the **3D Perspective** view, you must fully configure:

- Source
- Calibration Matrix
- Dimensions (X, Y)
- Origin

The **Position** and **Orientation** are necessary for the **CoP** (Center of Pressure) and **Force** vector to be represented correctly, but they can be changed according to the setup. In subsequent sessions, you may wish to configure additional properties to suit the needs of your motion capture application.

Change force plate display options

You can configure colors and other force plate display options in the **Options** dialog box (F7).



You turn a display option on or off by selecting or clearing the corresponding check box.

> **Tip**
>
> You can set colors for some of the display options. If a default color is listed as Context Color (this indicates the color is set to the system default) and you want to change it, click the down arrow to the right of the drop-down list, and clear the Macro check box. You can then activate the color picker.

To view force plates in a **3D Perspective** view, ensure the **Force Plates** check box on the left side of the **Options** dialog box is selected. You can change the following options:

Force Plate section

Property	Description
Draw Plate	When selected, the force plate appears in the View pane.
Draw Number	By default, each force plate displays a number (e.g., 1, 2, 3). The number corresponds to the order in which the force plates appear in the **System Resources** tree. You can turn off the number display by clearing the check box.
Draw Axes	You can configure how the force plate axes appear in the **3D Perspective** view by selecting the **Draw Axes** check box. The axes appear below the floor plane of the volume.
Axis Width	Width of the axis. Set the axis width by typing a value or by moving the slider.
Axis Length	Length of the axis. Set the axis length by typing a value or by moving the slider.

Foot Contact Colors section

If you have designated a force plate to be a left or right foot contact, by default Nexus displays a left-foot contact force plate in red in the **3D Perspective** view, a right-foot contact force plate in green. An auto-detect force plate is displayed in light gray, and an invalid force plate (neither left, right nor auto-detect) is displayed in dark gray.

To change the default colors for the force plate display, in the **Options** dialog box, select the **Force Plates** option, and in the **Properties** pane on the right, click on a color in the **Foot Contact Colors** section to activate the color picker.

Force Vector section

You can visualize the force vector by selecting the **Draw** and **Draw Butterfly** check boxes. Then you can set the quality of the vector display, as well as the radius and scaling factor (length), the butterfly style (whether the butterfly is represented as transparent, as a wireframe or both), and color.

Setting a **Scaling Factor** value causes the vector to be displayed as an arrow originating from the force plate in the direction of the force.

Values below the **Force Threshold (N)** are not visualized as force vectors on the plate. This setting does not affect the value for **Force Threshold** that is written into a recorded trial (see Visualize and record Force Threshold on page 83).

Moment Vector section

You can select whether the moment vector is visually represented in the **3D Perspective** view, as well as the quality, radius and color of the moment vector display.

You can see the results of your settings in the **3D Perspective** view as you configure an option.

Visualize and record Force Threshold

The Force Threshold of a force plate specifies the noise floor value in Newtons. Forces below this value are assumed to be noise and are clamped to zero.

An Options setting for force plates enables you to specify a different value for the Force Threshold depending on whether you are recording data or displaying a trial.

- The value for the Force Threshold that is set in the System Resources pane for the force plate affects the values that are written into a recorded trial. For example, if the Force Threshold is set to 25 N, all values below this are considered noise and values of zero are recorded.
- The value for the Force Threshold that is set in the Options dialog box ensures that values below this threshold are not visualized as force vectors on the plate. This prevents very small values, which are simply noise, from creating a distracting 'flickering' force vector on the plate. If you are distracted by flickering while viewing a trial, raise this value to remove the flickering. This does not affect the value for Force Threshold that is written into a recorded trial.

To specify the Force Threshold that is recorded for a force plate:

1. Ensure the system is in Live mode.
2. In the System Resources tree, select the force plate.
3. In the Properties pane, ensure the Advanced properties are displayed.
4. In the General section, change Force Threshold (N) from the default (25 N) to the required value.
   Values below this magnitude are ignored and are recorded as zero.

To specify a value for visualizing the force vector:

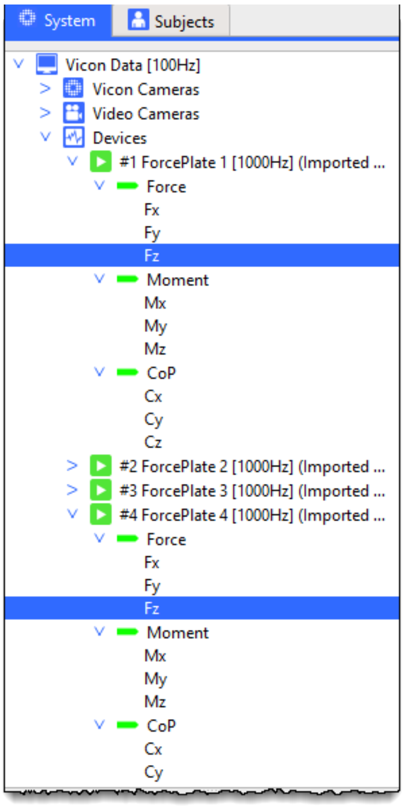
1. Press F7 to open the Options dialog box.
2. On the left click ForcePlates.
3. In the Properties pane on the right, in the Force Vector section, change the Force Threshold (N) to the required value (the default is 25 N).
   Regardless of the setting of the Force Threshold in the System Resources pane, the force vector that is displayed in the view pane is below the threshold that is set in the Options dialog box. For example, if you set the value to 10 in the System Resources pane, but left it at 25 in in the Options dialog box, a value of 15 would be prevented from causing flickering, but would be recorded in saved trials.

View combined output from multiple force plates

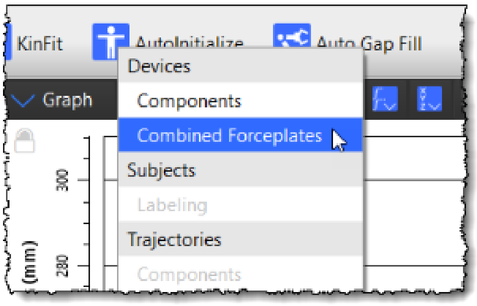
You can combine selected outputs from multiple force plates, both in Live mode and when reviewing captured trials in Offline mode. For example, in trials where the subject's feet land on separate force plates, you can examine the combined landing force or overall CoP from both feet.

To view combined output from multiple force plates:

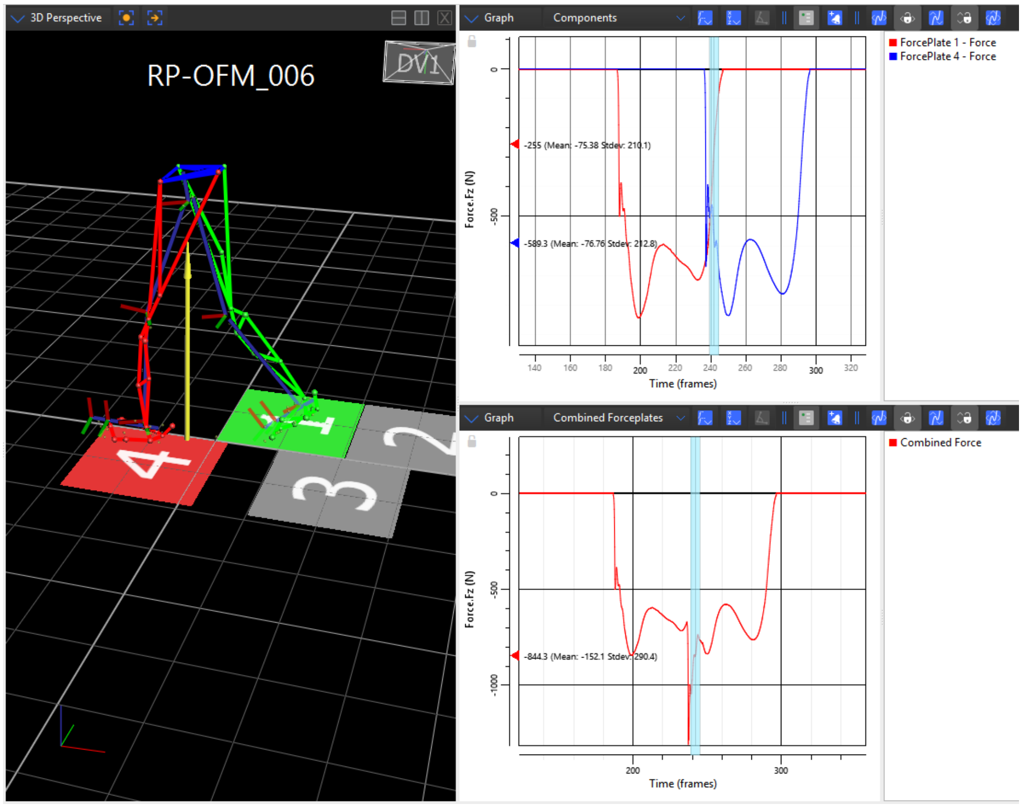
1. In the Options dialog box (press F7), on the left, click ForcePlates and in the Properties pane on the right, in the Force Vector section, ensure Draw Combined Force Vector is selected.
2. In the System Resources tree, CTRL+click to select the required outputs from each force plate.
   Ensure you select identical outputs from each force plate.



3. In a **Graph** view, in the **Graph type** list, select **Combined Forceplates**.



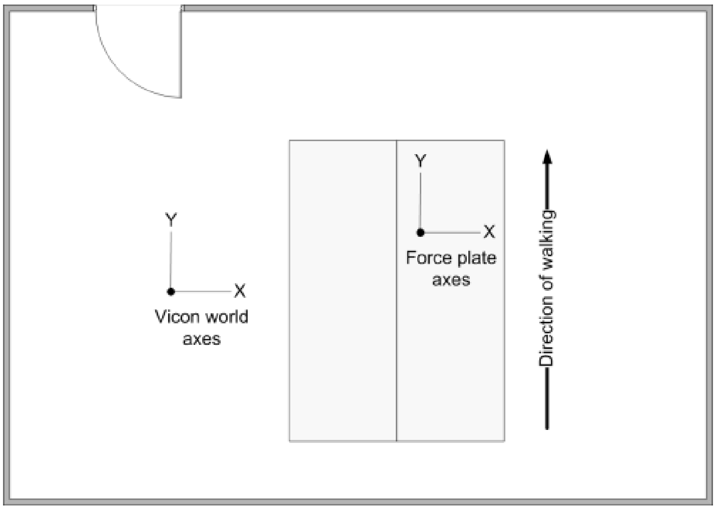
4. In the **Graph** view, the combined output you selected is displayed.



Usage recommendations for Motekforce Link treadmill

Note the following recommendations for setting up and using a Motekforce Link treadmill with Nexus:

- For ease of use, align the Vicon world Y-axis with the direction of walking on the treadmill. To achieve this, you can either move the treadmill in the laboratory to align with the Vicon world axes in Nexus, or change the Vicon world axes to align with the position of the treadmill. You can then set the incline of both plates of the treadmill by adjusting the **XOrientation** property of the force plate.



- Assuming you follow this advice so that the direction of walking on the force plates aligns with the Vicon Y-axis, and the left-right (medial-lateral) direction aligns with the Vicon X-axis, inclination of the force plate will correspond to a rotation of the plate about its X-axis. For example, to set a 10 degree inclination of the treadmill, you would set a 10 degree **X (deg)** rotation in the plate's **Orientation** properties.
- Each capture must use only one inclination of the treadmill. If you need to capture multiple inclinations, to enable you to switch easily between different inclinations, create a number of separate system files: one for each required inclination.

Configure EMG devices

EMG data can be acquired through the analog capture functionality of a Vicon connectivity device. If your EMG device has a digital output, it is possible that this data stream can also be captured, but this depends on the data stream's format and the equipment's manufacturer. For more information on the digital data streams that can be used with your Vicon system, contact Vicon Support.

To configure EMG devices for data capture:

1. If Vicon Nexus is not in Live mode, in the **System Resources** pane, click the **Go Live** button.
   The first time you use Nexus, the **Devices** node is empty. You must add and configure a new EMG device before it will be displayed in the **System Resources** pane.
2. In the **System Resources** tree, right-click **Devices**, point to **Add Analog Device** or **Add Digital Device** and from the context menu select the type of EMG device that is integrated in your Vicon system.
   The **Devices** node automatically expands to display the newly created EMG device with its attendant output:
   - An analog Accelerometer will have an **Acceleration** output
   - An analog EMG will have a **Voltage** output
   - A ZeroWire EMG will have a **Voltage** and a **Foot Switch** output
3. In the **System Resources** tree, select the node whose properties you wish to edit:
   - **Generic Analog Device** sub node for all devices
   - A sub node for a specific device

   The colored icon beside a device node identifies the status of the device and of any connected source:
   - Green play button: Component OK (active or connected); if an analog device is connected, the source is selected and all channels are configured.
   - Yellow pause button: Component is not fully set up (e.g., not all channels have been assigned a pin or the calibration matrix has not been configured).
   - Gray play button: Component connected but not contributing any data.
   - Red stop button: Component down (unavailable or disconnected).
4. In the **Properties** section at the bottom of the **System Resources** pane, view or change settings for the following properties:

a. In the General section:
   - Enter a Name to enable you to identify the device.
   - Set the Delay Compensation (in frames). All devices have a delay compensation value which adjusts the synchronization offset between the device and the Vicon data. The Delay Compensation slider bar enables you to correct this difference. Values can be set between -10 and 10 (the default is 0). Note that analog data collected with a Vicon Lock+ or an MX Giganet should already be synchronized, so in this case, this value should be set at 0.
   - Set the Amplifier Gain. The voltage gain scale factor. Can be set between 1-1000. The default setting depends on the device.

b. In the Source section, select a Source (the Vicon connectivity device to which the accelerometer or EMG is attached) from the drop-down list.

When you first set up your Vicon system, you must configure at least the Name and Source properties, add channels and designate the Source for each channel. In subsequent sessions, you may wish to configure additional properties to suit the needs of your motion capture application.

5. In the System Resources tree, expand the device node you added and select a device output sub node which corresponds to an output from the device (such as Acceleration for an accelerometer, or Voltage for an EMG device).
6. Right-click the device output and add an output component or components: For an accelerometer or analog EMG, choices are:
   - Add Component (or 2, 4, or 8 Components)

   For a ZeroWire EMG, choices are:

   Under Voltage:
   - Add EMG (numbered 1 - 16)
   - Remove Channels (if you have added channels)
   - Enable Auto Populate (or Disable Auto Populate if the AutoPopulate Properties box is selected)

   Under Foot Switch:
   - Add FSW (designated A1-A4 and B1-B4)

   A new component node (or nodes) is added to the tree.

   The colored icon beside the device output node identifies the analog channel status:
   - Green arrow: Analog channel connected to source device.

- Yellow arrow: Analog channel not fully set up (e.g., not all channels have been assigned a pin or the calibration matrix has not be configured).

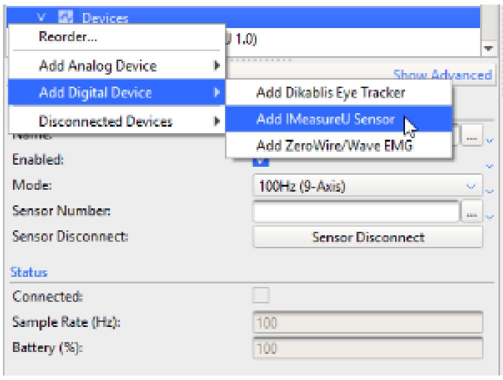
7. In the **Properties** pane under the **General** section, specify a **Name** for the node or nodes you have added.
   For the ZeroWire EMG, **Name** is the only property setting necessary.
8. In the **Properties Source** section:
   a. Assign the **Pin** to one of the available pins on **Source** device. After you have assigned a pin, the status icons will turn green.
   b. Specify the **Gain** (V).
   c. Set the **Zero Level**.
9. At the top of the **System Resources** pane, click the Save button to save your system configuration settings to a .system file in the **Systems Configurations** folder (see Manage configurations in Vicon Nexus on page 21).

For more information on the properties for EMG and other devices, see *System Resources nodes* in the *Vicon Nexus Reference Guide*.

Configure IMUs

To enable you to capture data from Vicon IMUs (Inertial Measurement Units) and optical markers simultaneously, you can use IMUs as digital devices in Nexus to stream and capture data. You can use IMUs to preview data in real time and to capture data onto the IMU's internal memory. You can then plug the IMUs into your PC via USB and use Nexus to download the captured data.

> **Important**
>
> Vicon IMUs are supported for use with Nexus for research purposes only. For full sensor safety and regulatory details, see the IMeasureU Sensor Safety and Regulatory Information, available from the Vicon website.



Before you begin using IMUs with Nexus, note the following points:

- Windows 10 and above only is supported. Ensure the latest Windows 10 updates are installed.
- To use Vicon IMUs in Nexus, first plug the supplied Bluetooth® (BLE) dongle into the relevant Windows PC. Ensure that its drivers have been updated: you can do this using the Windows Device Manager.
- To enable further setup and control of your IMUs after they are paired and recognized by Windows 10, you can download IMU Lightning desktop software. IMU Lightning enables you to see the status of IMUs outside of Nexus and is useful in certain troubleshooting situations.
- IMUs have five collection modes: 100 Hz (9-axis), 250 Hz (6-axis), 500 Hz (6-axis), 500 Hz (9-axis), or 1000 Hz (3-axis). To accommodate the IMUs, ensure that your Vicon system is running at a compatible frame rate.
- Nexus supports the use of up to four current Vicon IMUs.
- You can transfer the data that is collected and stored on your IMUs onto your PC using the **IMU Transfer** pane in Nexus (see Transfer data from IMUs on page 227).
- To enable transfers to work, ensure you have downloaded and installed the CP210x USB to UART Bridge VCP Drivers.

Set up IMUs for capture

The following are brief notes on working with IMUs with Nexus. For more information on setting up and using IMUs to stream and capture data, see the IMeasureU documentation.

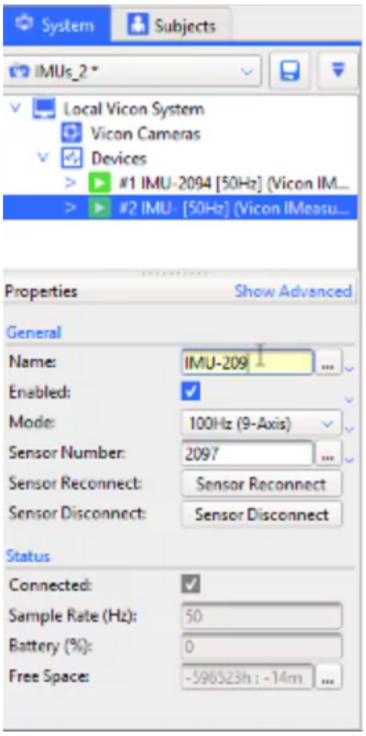
For more detail on working with IMUs and Nexus, watch the Vicon videos, IMU Unboxing, Introduction to using Vicon IMU with Nexus, and Nexus and the IMU Research app, available on YouTube.

1. If necessary (see previous points on page 90), download and install IMU Lightning.
2. Ensure that the Bluetooth® dongle is plugged into the PC and that its drivers are up-to-date.
3. Turn on the IMUs and pair them to the relevant Windows computer, via Bluetooth settings in Windows 10.
4. In Nexus, add the IMUs as digital devices and in the **Properties** pane, specify the required settings:
   a. In the **System Resources** pane, expand the **Local Vicon System** node, then right-click the **Devices** node, point to **Add Digital Device** and click **Add IMeasureU Sensor**.
   b. In the **Properties** pane below, specify a **Name** and **Sensor Number**:

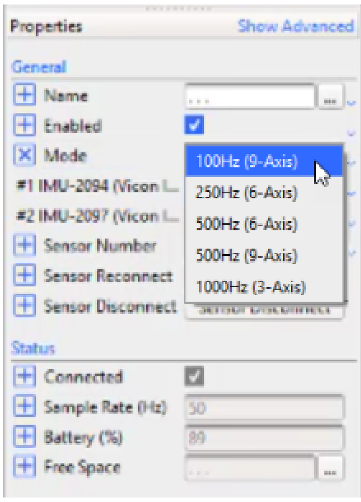
Property	Description
Name	The identifier that you want to give to the IMU.
Sensor Number	The unique serial number of the device, which is found on the tag on the side of the IMU. Although you enter only the digits on the tag, this is an eight-digit number that is padded with four zeros when displayed in the Log.

      For more information on the available properties, see IMeasureU Sensor node in the *Vicon Nexus Reference Guide*.
   c. Note that if the connection to the IMU is lost (the device is displayed as gray in the **System Resources** pane), you can click the **Sensor Reconnect** button to reconnect. This button is not available while you are capturing a trial.

   When you have specified a name and number, in the Resources list, the Play icon next to the IMU name turns green.



5. If necessary, change the collection mode (frequency) at which the IMUs are to run.
   To do this:
   a. In the Resources list, ensure the IMUs are selected.
   b. To display the axes for the IMUs, in the view pane, select Graph view.
   c. In the Properties pane, from the **Mode** list, select the required frequency.



   d. In the Graph view, the graphs change to reflect the selection.

> **Note**
>
> If you need to calibrate the IMUs (normally only necessary when you update the camera firmware or if an offset occurs), see Calibrate IMUs on page 231. Note that, for calibration, you must set the collection mode to 100 Hz.

6. Attach the IMUs to the subject.
7. Set up and start capture in the usual way (see Capture the required movement on page 137 in the *Vicon Nexus User Guide*).

For more information, see Work with IMUs on page 222.

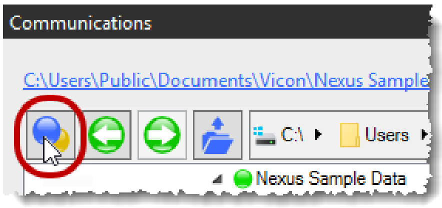
Prepare a data storage location

Before you capture and process a trial, you must create a hierarchical structure for storing and managing all the data associated with the motion capture trial. You do this in the **Communications** pane, on the **Data Management** tab.

For a video guide to database management, see the Vicon video, proEclipse: Preparing and managing your database, which is available on YouTube.

To prepare a location for your trial data:

1. In the **Communications** pane, click the **Data Management** tab.
2. On the toolbar, click the **Show main proEclipse menu** button.

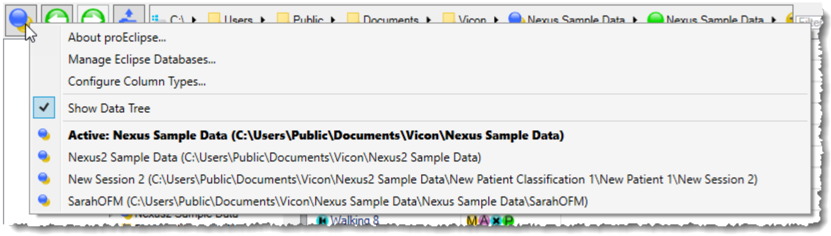


3. Click **Manage Eclipse Databases**, and in the **Manage Eclipse Databases** dialog box, click the **Add New** button to enable you to locate and register a database folder. (Registering a folder enables you to select it in the **Currently Registered Eclipse Databases** list and from the recently used files that are displayed when you click the **Show main proEclipse menu** button.)

> **Note**
>
> A database is just a folder structure where your Nexus files will be saved. Generally, you can create a database in any unrestricted location.

4. Select a location on your local hard disk for the database folder, or if required, click **Make New Folder** and enter a name for your folder. Click OK.
5. In the **Manage Eclipse Databases** dialog box, the folder that you just selected or created appears in the **Currently Registered Eclipse Databases** list. Click **Close**.

All registered databases are also available when you click the **Show main proEclipse menu** button.



6. On the **Data Management** tab, click on name of the folder you just created, ensuring its name appears in the live link at the top left of the tab.

> **Tip**
>
> To check that your hierarchy of folders appears as expected, click the live link to open an instance of Windows Explorer, where you can see the folder you created.

7. Ensure the folder name is still selected in the hierarchy displayed on the left of the **Data Management** tab, and create a hierarchy of data folders in which to store your data. A good practice is to make sure all your data goes into a session folder. To do this:
   a. Add a top-level folder by clicking the green **New Patient Classification** button . You can change its name as required. Generally, this classifies a group of subjects such as Men/Women, Old/Young, or Target Population /Controls.
   b. In the left hand panel, click the new patient classification that you just added and then add a patient folder by clicking the yellow **New Patient** button . Again, you can rename it as required.
   c. In the left-hand panel, click the patient folder that you just added and then add a session folder by clicking the gray **New Session** button . Rename it as required.
8. Select the new session folder by clicking it on the **Data Management** tab.

You now have an active session, ready to store your preliminary capture.

For more detailed information on data management, see *Data management with Nexus* in the Vicon Nexus Reference Guide.

Calibrate a Vicon system

The following topics cover calibrating the Vicon system, including the Vicon cameras and any video cameras, for motion capture.

After you have configured your system (see Prepare a Vicon system on page 43), the next setup step is to calibrate the Vicon cameras and define the origin of the capture volume. Calibration enables Nexus to determine the positions, orientations, and lens properties of all the Vicon cameras and to produce accurate 3D data.

Important

Vicon recommends that you calibrate your Vicon cameras after any changes to your system and each day, before you capture any data. This ensures that any accidental changes in your setup (such as a camera being knocked while the system was unsupervised) does not influence the quality of your data. You can perform the level of camera calibration that suits your requirements: a full camera calibration or a calibration refinement.

For more information see:

- Calibrate Vicon cameras on page 97
- Set the volume origin on page 101
- Calibrate the floor plane on page 103
- Manage camera calibrations on page 105

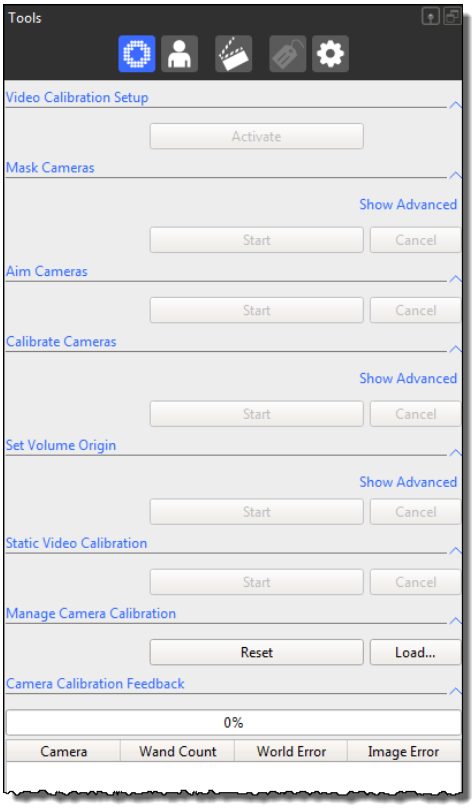
A Vicon Nexus 2 Calibration video showing these procedures is available on YouTube.

Note

As the videos were recorded using an earlier version of Nexus 2, you may notice small differences in the user interface.

Calibrate Vicon cameras

To calibrate Vicon cameras, you use the relevant sections of the **System Preparation Tools** pane.



To perform a camera calibration, you need a Vicon calibration device, which is supplied with your Vicon system. Note that, to use the following process to calibrate supported digital video cameras, as well as optical cameras, you must use an Active Wand, which must be set to Strobe mode. (If you are not using an Active Wand, you can calibrate any supported video cameras using the legacy **Static Video Calibration** section of the **System Preparation Tools** pane. For more information, contact Vicon Support.)

Camera calibration describes the capture volume to the system, enabling Nexus to produce accurate 3D data. During camera calibration, Vicon Nexus creates a calibration parameters (*.xcp*) file. This file contains the calibration settings and threshold data specified for the Vicon cameras (as well as any supported video cameras) in your Nexus system and is used when data from these cameras is processed.

> ⚠ **Important**
>
> Vicon recommends that you calibrate your Vicon cameras each day before you capture any data. This ensures that any unexpected changes in your setup that may have occurred when the system was unsupervised (such as someone accidentally knocking a camera slightly out of position) will not influence the quality of your data. You can perform the level of camera calibration that suits your requirements: a full camera calibration or a calibration refinement.

To calibrate Vicon cameras in Nexus:

1. Ensure you have aimed and masked the cameras (see Aim Vicon cameras on page 64 and Mask unwanted reflections on page 66).
2. Remove from the capture volume all markers and the sources of any unwanted reflections that have not been accounted for by camera masks previously created in a **Camera** view.
3. In the **Resources** pane, ensure Nexus is in Live mode.
4. Display a **Camera** view.
5. In the **System Resources** tree, expand the **Vicon Cameras** node and select all Vicon cameras.
6. In the **System Preparation Tools** pane , expand **Calibrate Cameras** and from the **Wand** list, select the calibration device that you are using.
7. If your Vicon system includes video cameras, in the **Video Calibration Setup** section, click **Activate** to start video calibration mode. If your system does not include video cameras, go straight to the next step.
   If you have not already saved the current configuration, you will be prompted to save it. Ensure its name is recognizable as a calibration and then choose whether it will be **Shared** or Private (see Manage configurations in Vicon Nexus on page 21).

   > ✔ **Tip**
   >
   > The settings for some video camera properties may differ in video calibration setup mode from those in live capture mode. For these properties, any changes made after you click **Activate** only apply during video calibration setup mode. The properties revert to their live capture settings after you click **Deactivate**.

8. In the **Calibrate Cameras** section, view or change settings for the required parameters:
    a. Select the appropriate **Calibration Type: Full Calibration** or **Calibration Refinement**.

    > **Tip**
    >
    > If you have not already calibrated all cameras, you cannot perform a calibration with the **Calibration Refinement** option. For more information on calibration refinement, see Understand camera calibration refinement in the *Vicon Nexus Reference Guide*.

    b. To automatically stop calibration when sufficient information has been acquired, ensure **Auto Stop** is selected.
9. In the **Calibrate Cameras** section, click **Start**. The camera calibration process starts, and the **Start** button switches to its **Stop** setting.
10. In the capture volume, wave the calibration wand throughout the area where you intend to capture 3D data, ensuring that the markers on the calibration object are visible to the cameras. Vicon Nexus begins to capture wand wave data.
11. Monitor the calibration progress and status:
    - In each **Camera** view, ensuring that the colored lines that identify wand frames indicate that sufficient wand frames are spread across the intended 3D capture volume. Also check the colored triangle in the lower right of each view pane. The triangles change from red to green as the collection of calibration information progresses, and then vanish when the cameras are calibrated.

    > **Tip**
    >
    > If no cameras are visible in the **Camera** view, ensure that the **Show Uncalibrated Cameras** option is selected in the **Camera Positions** section of the **Options** dialog box (press F7). Also, if you are not using the Vicon Control app, you may find it helpful to display the Nexus window on a large screen where the person in the capture volume can see the feedback in the **Camera** view while they are waving the calibration wand, so they can see the area that they have covered.

    - In the **System Preparation Tools** pane, under the **Camera Calibration Feedback** section check **Wand Count** values returned for each camera.

- On the strobe unit on the front of each camera, the status light flashes while the camera registers valid frames where the whole wand is visible. When the status light turns a solid green, the camera has collected enough data to be calibrated successfully, usually 1,000 frames of valid wand data.

12. If **Auto Stop** was not selected, in the **Calibrate Cameras** section, click **Stop**. Nexus starts processing the calibration information.
    If you selected the **Auto Stop** option, Nexus automatically stops collecting calibration information and starts processing it when sufficient calibration information has been acquired.
    Note that, for a system that includes video cameras, calibration is done in two passes, so the **Camera Calibration Feedback** bar progresses from 0% to 100% twice.
13. In the **Camera Calibration Feedback** section, monitor the progress bar until the camera calibration process is complete and review the **Wand Count** and **Image Error** data. As a general guideline, Nexus typically takes 15–60 seconds to complete its calculations for a typical Vicon system setup, but this can take longer for a very large system or low-specification PC. When the camera calibration has successfully completed, it is automatically saved to an *.xcp* file. For more information on the controls in this section, see Camera Calibration Feedback section in the *Vicon Nexus Reference Guide*.

    Tip

    Because calibration feedback values are based on factors such as the size of the capture volume and the camera lens type, it is not possible to provide general guidelines on typical or acceptable ranges. Therefore, to determine the optimal values for your Vicon system, shortly after the system is installed, establish a baseline against which you can compare future daily calibration values.

14. If you have calibrated video cameras, to exit video calibration mode and return to your settings for live captures, click **Deactivate** in the **Video Calibration Setup** section.

When you have finished calibrating the Vicon cameras, you are ready to Set the volume origin on page 101.

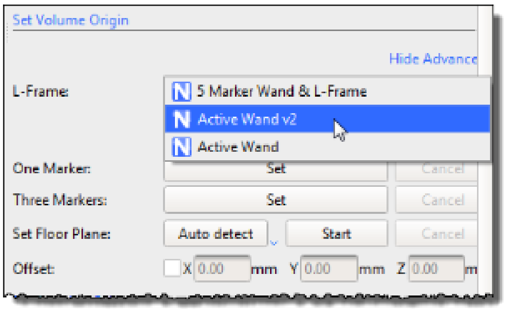
Set the volume origin

Setting the volume origin (global coordinate system) tells the Vicon system where the center of your capture volume is and what its orientation is (x, y, and z axes), so that subjects are displayed the right way up in the Nexus view pane and so that you can change the way data is visualized. You normally set the volume origin immediately after the previous step Calibrate Vicon cameras on page 97.

To set the volume origin, you need a calibration device, normally an Active Wand, which is supplied with your Vicon system.

To set the global coordinate system:

1. Remove from the capture volume all markers and the sources of any unwanted reflections that have not been accounted for by camera masks previously created in a Camera view (see Mask unwanted reflections on page 66).
2. In the Resources pane, ensure Nexus is in Live mode.
3. Display a 3D Perspective view.
4. In the System Preparation tools pane, expand the Set Volume Origin section and from the L-Frame drop-down list, select the type of calibration device you are using to set the volume origin. (This will normally be the calibration device that was supplied with your Vicon system.)



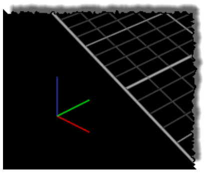
If it is impractical to place the wand in the required position, you can instead use the One Marker or Three Markers options to set the origin. (The Three Markers option creates two vectors that determine the orientation of the volume.) To use these options, position the marker(s) in the volume and then click Set next to the chosen option. You can then select the required marker(s) in the 3D Perspective view. If you selected Three Markers, you are prompted to select an origin marker, a primary (X-axis) marker, and secondary (Z-axis) marker.

5. In the capture volume, place the calibration object flat on the floor in the position and orientation that you would like to be the origin of the global coordinates system (often a force plate corner or another clearly marked area of the volume).

6. In the **Set Volume Origin** section, click **Start**.
   The calibration object tracking process starts, Nexus identifies the calibration object in the capture volume, displays a 3D representation of it in the **3D Perspective** view, and switches the **Start** button to its **Set Origin** setting.

7. Click **Set Origin** to complete the calibration object tracking process. Nexus sets the global origin and axes to correspond to the position and orientation of the calibration object in the capture volume. In the **3D Perspective** view, the floor grid is displayed aligned with the capture volume floor and the representations of the cameras are distributed in the position and orientation in which the physical cameras are located around the capture volume. When the global coordinate system has been successfully set, it is automatically saved to an .xcp file. The latest calibration (.xcp) file is saved to the default location. This calibration file is automatically copied to every subsequent trial captured.

8. Verify that the global coordinate system was successfully set by checking that the system tracks the calibration object.
   If it does not, check that you selected the correct calibration object from the **L-Frame** drop-down list at the top of the tools pane. If not, repeat this procedure from step 4, ensuring that you select the correct entry for the calibration device you are using. After you have set the global coordinate system, you can display the volume axes marker in the bottom corner of the **3D Perspective** view.

9. Turn the display of the volume axes on or off in the **Options** dialog box (F7) by selecting or deselecting **Volume Axes** under **General View Options**.

Calibrate the floor plane

To ensure that your coordinate system is accurately aligned with the floor of the capture volume, which is particularly useful if the floor is uneven, you can adjust the position of the floor plane using markers in the volume to automatically define it.

To calibrate the floor plane:

1. Complete the camera calibration procedure (see Calibrate Vicon cameras on page 97) and set the origin (see Set the volume origin on page 101).
2. Turn off the calibration object or remove it from the volume.
3. Scatter a quantity of the same size Vicon markers across the floor of your capture volume. If you want Vicon Nexus to automatically recognize these markers as floor plane markers, ensure that they outnumber any other groups of markers (e.g, markers on a wand or markers that were used to focus cameras, etc).
4. In the **System Preparation Tools** pane, expand the **Set Volume Origin** section, click **Show Advanced**, and ensure that the options relating to **Set Floor Plane** are as required:
   - **Auto detect**: Markers are automatically detected, based on the marker group that defines the plane with the most markers. (If, when you click **Auto detect**, the wrong markers are selected in the **3D Perspective** view, you can add or remove markers in the volume until selection is as required.) To ensure that the tolerance setting (in mm), which specifies the height of the group of markers that define the floor plane, is as required, click the drop-down arrow next to **Auto Detect**, and set the tolerance to a value that prevents a large variance in height (2mm is often enough).
   - **Start**: Enables you to manually select the markers that will determine the floor plane by clicking the required markers in the **3D Perspective** view.
   - **Offset**: Select the check box and set the amount (in mm) by which to adjust the floor plane (default is zero) in X Y and Z planes. Because Nexus finds the centers of the markers, set a Z-offset that accounts for the size of the markers plus their bases (for example, for 14mm markers on 2mm bases, the Z-offset is -9mm).
5. If you clicked **Start**, when you have selected the required markers, click the **Stop** button.
   In the **3D Perspective** view the cameras shift as a group slightly along one or

more rotation axes to better reflect an average of the markers scattered across the floor, taking into account any offsets that you specified. The latest calibration (.xcp) file is updated. This calibration file is automatically copied to every subsequent trial captured.

Your Vicon cameras are now calibrated and ready to capture data.

Manage camera calibrations

In the **Manage Camera Calibration** section of the **System Preparation Tools** pane, you can click **Auto number cameras** to automatically number the cameras number in ascending order, according to their position in the capture volume.

The **Manage Camera Calibration** section also enables you to reset or load camera calibrations that define settings for Vicon cameras.

- Automatically number cameras on page 105
- Reset and load camera calibrations on page 107

Automatically number cameras

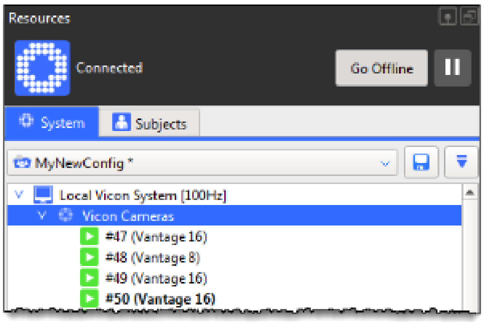
The **Auto number cameras** button enables you to quickly number the currently connected Vicon cameras in ascending order, according to their position in the capture volume. To obtain useful autonumbering, you normally auto-number the cameras at some point after you have aimed (and calibrated) the cameras.

> **Tip**
>
> You can auto-number the cameras at any point after you have performed the **Aim Cameras** operation (see Aim Vicon cameras on page 64): you do not need to perform a complete camera calibration to auto-number the cameras.

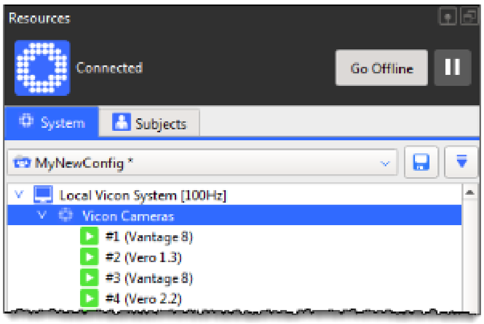
Automatic numbering starts with the camera that is furthest from the volume origin. The cameras are then numbered in a clockwise direction around the volume. If your cameras are positioned at different levels, the cameras in the level that contains the most cameras are numbered first.

To automatically number Vicon cameras:

1. Ensure Nexus is in Live mode and that you have aimed the cameras.
2. On the **System Resources** tab, ensure you can see a list of Vicon cameras.



3. On the **System Preparation Tools** pane, in the **Manage Camera Calibration** section, click **Auto number cameras**.
   The cameras are automatically numbered in ascending order, according to their position in the volume.



4. In the volume, check that the cameras are now numbered as required.

Reset and load camera calibrations

During camera calibration, Vicon Nexus creates a calibration parameters (.xcp) file. When you change the currently loaded .xcp file, either by calibrating cameras or by setting the global coordinate system, Nexus stores the calibration state before the changes. This enables you to revert to the previous calibration or load a different calibration at any time while using Nexus (for example, you would normally want to load a different calibration when you connect your Nexus PC to a different system).

Changing a camera calibration can be useful in the following circumstances:

- To undo a poor calibration
- To compare calibration changes

To manage camera calibration files in Nexus:

In the System Preparation Tools pane, under the Manage Camera Calibration section, click the desired button:

- Reset Removes all non-existing cameras, clears the calibrated position for existing cameras, and reverts all calibration parameters to their default settings. This enables you to recalibrate the system from a clean starting point.
- Load Enables you to load a previously saved calibration file. In the Choose a file dialog box, navigate to and select the desired camera calibration (.xcp or .cp file) and click Open.

If a camera is not calibrating well or has been bumped after calibration, you can remove it from the current calibration prior to data capture.

To remove a camera from an existing system calibration:

Use any of the following methods:

- In the System Resources pane, right-click on the node for the camera you want to remove and select Reset Calibration; or
- In the System Resources pane, select the node for the camera you want to remove and in the Properties pane, scroll down to the Calibration section and click Reset Calibration; or
- In a 3D Perspective view, right-click the camera that you want to remove and then click Reset Calibration.

Prepare a subject

After you have configured and calibrated your Vicon system, and prepared a data management hierarchy in which to store your data, you can prepare the subject whose motion is to be captured.

To start with, you may find it easiest to use the standard Plug-in Gait marker set and in Vicon Nexus, create a subject that is based on one of the Vicon-supplied labeling skeleton templates (VSTs), as described in the following topics. If you are planning on processing your captured data with Plug-in Gait, then you must use one of the supplied Plug-in Gait VST files. For more information, see Modeling with Plug-in Gait on page 185
.

Later, if required, you may want to create your own custom labeling skeleton template. The labeling skeleton template that you use is determined by your particular application and your marker set. For information on how to create a custom labeling skeleton template, see *Creating labeling skeleton templates (VSTs)*.

Having chosen your marker set, you can prepare the subject data in Nexus. To do this, you use the **Subjects Resources** pane and the **Subject Preparation Tools** pane to create a new subject from a template and to calibrate the labeling skeleton. These and other procedures that you may find useful while creating and calibrating your labeling skeleton are described in the following topics:

- Create a new subject from a template on page 110
- Calibrate a labeling skeleton on page 113
- Correcting swapped labels on page 127
- Manually label a trial on page 128
- Work with pipelines on page 131

If you have problems calibrating your labeling skeleton, also see Troubleshooting labeling skeleton templates in *Creating labeling skeleton templates (VSTs)*.

Important

A Nexus .vst file is used only to define the marker set and to enable Nexus to perform automatic labeling. It is not a biomechanical model that will output valid joint angles or other kinematic/kinetic variables. To derive valid kinematics or kinetics, use either a predefined model (such as Vicon Plug-in Gait, as described in this documentation) or create your own model with Vicon BodyBuilder, MATLAB or Python.

Create a new subject from a template

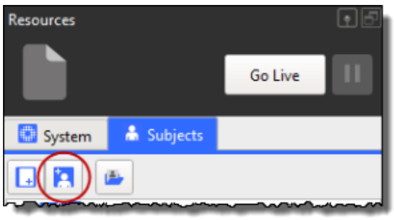
Before you calibrate a labeling skeleton, you must create a new subject node for your subject. The following instructions explain how to create a new subject node in the **Subjects Resources** tree, and base it on an existing Vicon labeling skeleton template (.*vst* file) in the **Subjects Resources** pane.

Sample .*vst* files, including those for Plug-in Gait, are provided in the Nexus model templates folder (by default, *C:\Program Files (x86)\Vicon\Nexus2.#\ModelTemplates*). If you are using a standard Vicon marker set, you can base a new **Subject** node on one of these templates. You subsequently scale the template to fit your particular subject when you calibrate the labeling skeleton (.vsk file).

If you are not using a standard marker set, or need to modify a standard set, you will need to create a custom labeling skeleton template. For more information, see *Creating labeling skeleton templates (VSTs)* or contact Vicon Support.

To create a new subject based on a Vicon labeling skeleton template:

1. Ensure that you have created or opened a session in a database in which you want to store the trial data, and that it is the active session (see Prepare a data storage location on page 94).
2. In the **Subjects Resources** pane toolbar, click the **Create a new Subject from a Labeling Skeleton** button.

   A list of all Vicon labeling skeleton templates (.vst files) currently contained in the Nexus ModelTemplates folder as well as a **Browse** link are displayed.

   > **Tip**
   >
   > The supplied templates, whose names end in Ai, enable you to autolabel static frames.

3. Select or browse to the desired .*vst* file. (For details of Plug-in Gait templates, see *Plug-in Gait models and templates* in the *Plug-in Gait Reference Guide*.

4. In the **Enter Subject Name** dialog box, specify the name for your new subject (of no more than 32 characters) and click OK.
5. If you used the **Browse** link rather than selecting a *.vst* file from the list, the **Choose a Subject** file dialog box is then displayed. Navigate to and select the *.vst* file on which the Vicon labeling skeleton (.vsk file) is to be based.
   The new subject node is added in alphabetical order to the **Subjects Resources** tree. The node automatically includes any sub nodes and data for the elements defined in the selected .vst file: **Markers**, **Segments**, **Joints**, and **Model Outputs**.

   > **Tip**
   >
   > Marker names are displayed in gray if they are defined in the template but the markers are not yet labeled.

6. In the **Properties** pane for the newly created subject, enter values for all the measurements that are required for this template. For information on the measurements required to run the supplied Plug-in Gait model, see Take subject measurements for Plug-in Gait on page 191 .

   > **Tip**
   >
   > Required measurements are highlighted in pink until you supply a value for them; optional measurements are not highlighted. Note that these measurements are required to run the relevant model; they do not affect labeling.

7. In the **Subjects Resources** pane, right-click the subject name and click **Save Subject**.

> **Note**
>
> When you save a labeling skeleton (VSK), Vicon Nexus automatically creates an accompanying *.mp* file, which is saved into the same folder as the VSK. If required, you can import this data by running an **Import MP** pipeline operation. This is only necessary in particular circumstances (for example, if you are using Vicon BodyBuilder or Vicon Polygon), and is not part of the normal Nexus workflow.

You can now place the appropriate marker set on your subject, (for information on attaching the Plug-in Gait marker set, see Attach Plug-in Gait markers to a patient on page 199) and calibrate its labeling skeleton (.vsk file).

Calibrate a labeling skeleton

Calibrating a Vicon labeling skeleton (VSK) enables Nexus to recognize the subject and automatically determine which labels belong to its reconstructions.

Having chosen a labeling skeleton template (VST) for your subject (see Create a new subject from a template on page 110), you then capture a short subject calibration trial (see below), run the required calibration operations on it and save the trial and the labeling skeleton (VSK). The calibration process applies the general information contained in the VST to your particular subject, who is wearing the marker set described in the VST. When you save the calibration, a labeling skeleton (VSK), containing information that is specific to your particular subject is saved.

You can then use that VSK to automatically label all the trials in an entire capture session. You only need to recalibrate if the subject changes, for example, if a different subject wears the same marker set, or if the markers are moved. For more detailed information about VSKs and labeling skeleton templates (VSTs) see *Creating labeling skeleton templates (VSTs)* .

Nexus provides a number of processes and multiple calibration options for calibrating a labeling skeleton. The factors that affect which process and which options are best for your situation, include:

- Whether your subject is able to perform a full ROM (Range of Motion) trial
- The type of movement you are studying
- Whether obtaining results quickly or labeling accuracy is more important

The following types of trial can be used to calibrate a labeling skeleton:

- ROM trial In a ROM trial, the subject completes a series of movements that exercises all of their joints. To produce the best calibration (and auto-labeling), the subject moves all of their joints through as full a joint range as possible. For the best results, ensure that each joint moves through a range that represents what the subject is likely to do during capture of trial data (ie, movement trials). Full ROM calibrations provide the most information of any of the calibration methods and therefore often provide the best labeling. For this reason Vicon recommends using a full ROM calibration whenever possible.
  For step-by-step instructions on using this type of trial to calibrate a labeling skeleton, see Calibrate a labeling skeleton using a ROM trial on page 115.
- Static trial This is a short trial where the subject stands in a base pose. A base pose is a static pose that is used when the VST (labeling skeleton template) file is first created. For Plug-in Gait, this is a T-pose or 'motorbike' pose. (To view images of the

motorbike pose, see Calibrate a labeling skeleton using a ROM trial on page 115.) This operation requires the least processing time, but provides less information than functional calibration (which is often used as part of ROM calibration, described above), as it operates on only a single frame of data. Consider using this type of calibration if:

- A subject's ability to perform a ROM trial is limited; and/or
- You want to obtain results as quickly as possible.

For step-by-step instructions on using this type of trial to calibrate a labeling skeleton, see Calibrate a labeling skeleton using a static trial on page 125.

- **Movement trial** This trial type is similar to a ROM trial, but focuses on only joint movements that are expected and specific to the activity being studied. For example, a gait laboratory might use a walking trial for this type of calibration. Movement trials are not normally used for an initial or full subject setup, but can be used in combination with some pipeline operations to improve labeling. They may be of benefit if the motion of the subject during movement trials is not similar enough to what is captured when using a static or ROM trial. For more information on this type of labeling skeleton calibration, see *Labeling skeleton calibration in detail* in the *Vicon Nexus Reference Guide*.

Whichever type of calibration you choose, note the following requirements for your calibration trial:

- Aim to produce a trial (ROM, static, or movement) that is as fully labeled as possible.
- The trial must contain only raw reconstructions; leave any gaps unfilled. (Unlabeled reconstructions have no influence on subsequent operations.) However, if necessary, you can manually label any missing markers (see Manually label a trial on page 128) or correct any swapped markers (see Correcting swapped labels on page 127).

For information on creating custom labeling skeleton templates (VSTs), see *Creating labeling skeleton templates (VSTs)* .

> **Note**
>
> This topic assumes that you are using a Vicon-supplied PlugInGait Ai VST, although you could instead use a custom VST, providing it conforms to the requirements listed in Requirements for custom labeling skeleton templates in *Creating labeling skeleton templates (VSTs)*.

Calibrate a labeling skeleton using a ROM trial

The recommended workflow for Nexus 2, which is described below, is to use a ROM trial and dynamic calibration to calibrate a labeling skeleton, as this is likely to produce more accurate automated labeling. However, if you need to use a static frame to calibrate your labeling skeleton (for example, if your subject cannot perform a full ROM, or you are capturing simple movement and/or want to obtain some results quickly), see Calibrate a labeling skeleton using a static trial on page 125.

After you have prepared a subject for capture (see Prepare a subject on page 108), to enable Nexus to provide correct automatic labeling for your subject, you capture a ROM (Range of Motion) trial that starts with a few frames where the subject is static, as described in the following steps.

Vicon Nexus then uses the information from the ROM trial to automatically label the subject in subsequent movement trials.

The following workflow is just one approach that you can adapt to suit your particular circumstances. For a detailed explanation of all the methods of subject calibration, see Labeling skeleton calibration in detail in the *Vicon Nexus Reference Guide*.

The workflow for calibrating a labeling skeleton based on a ROM trial can be divided into the following stages:

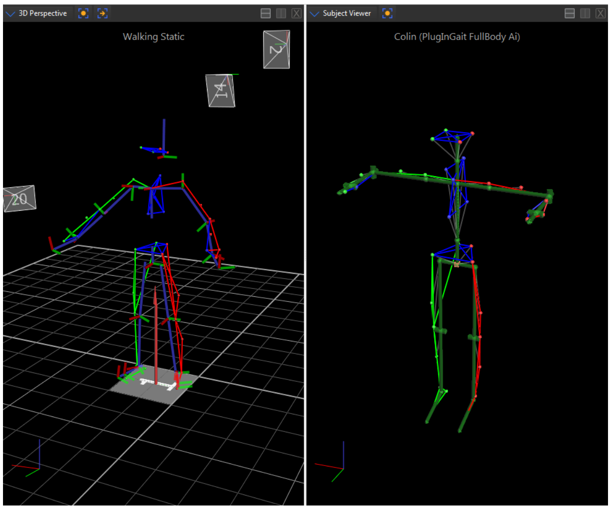
Perform an initial calibration of a labeling skeleton using a ROM trial

1. Ensure you have obtained any subject measurements required to run the model (required measurements are highlighted in pink in the Subjects Resources pane): for Plug-in Gait Lower Body, these are the body weight and height as well as measurements of both left and right leg lengths, knee widths, and ankle widths. These are used to calculate joint center positions, and must be measured and entered into Nexus before any modeling can take place can begin. Optionally, you can also measure tibial torsion, the inter-ASIS distance, the ASIS-to-greater trochanter distance, and the sole delta if the subject is wearing shoes. For more information on the measurements needed to run the Plug-in Gait model, see Take subject measurements for Plug-in Gait on page 191.
2. In the **Communications** pane, on the **Data Management** tab, ensure that you have created or opened the required database in which to store the data you will be capturing and that the required session is active. (For a reminder of how to do this, see Prepare a data storage location on page 94.)
3. In the **Resources** pane, ensure Nexus is in Live mode.

4. In the **System Resources** tree, select the **Local Vicon System** node and then in the **Properties** section under **General**, check that the **Processing Level**, which sets the amount of real-time processing that is performed on the source data, is set appropriately. If you are not sure, set it to at least the **Labels** level.
5. Display a **3D Perspective** view.
6. In the **Subjects Resources** tree, ensure the required PlugInGait Ai (Auto Initialize) labeling skeleton template (VST) file is specified for the subject (see Create a new subject from a template on page 110) and that the subject node you created from the template is the only entry enabled for capture. (When enabled, the check box contains a check mark.)
7. If required by your model, in the **Properties** section at the bottom of the **Subjects Resources** pane, enter any subject measurements you obtained in step 1.
8. In the **Subject Preparation Tools** pane, ensure that the subject you created is selected in the **Subject** list.

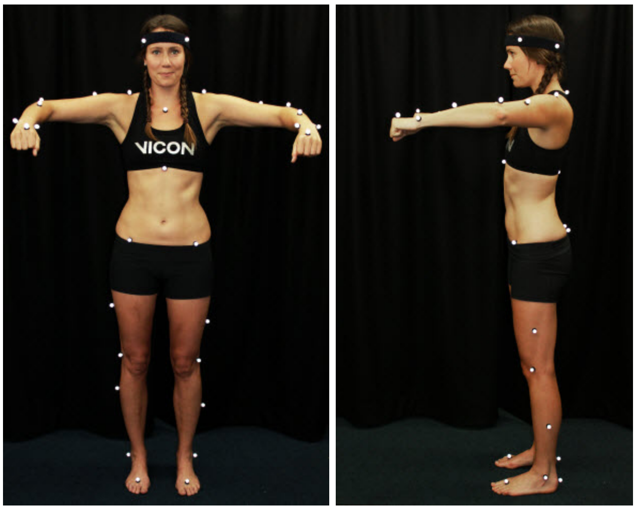
> **Tip**
>
> If you want to capture video or if you want to avoid automatically switching to **Offline** mode as soon as you have captured a trial, instead of clicking the **Subject Preparation** button, click the **Capture** button and on the **Capture** tab, ensure the settings are as required (for example, to capture video, ensure that in the **Data Source Setup** section, **Video Camera Data** is selected).

9. To enable Nexus to automatically label the key trajectories, in the capture volume, have the subject stand in the stationary neutral pose that was selected when the labeling skeleton template was created. If you are using a Vicon-supplied PiG template, ensure the pose is as described in step 12.

> **Tip**
>
> To help you to ensure that the subject adopts the correct base pose, on the view pane toolbar, click the **Split vertically** button and in the second pane, open a **Subject Viewer**. This displays the base pose for the current labeling skeleton template (VST).
>
> 
>

10. To make sure all the markers are visible, count the number of markers under the **Markers** node in the **Subjects Resources** tree, and make sure the same number of markers is visible in the **3D Perspective** view.
11. On the **Capture Tools** pane, in the **Capture** section, click **Start**.

    The **Start** button switches to its **Stop** setting.
12. Capture a ROM trial where the first 1–3 seconds are of the subject in the 'motorbike' pose, as shown in the following image, in which:
    - Arms are outstretched, held level or slightly lowered, with elbows bent and further forward than the shoulders (so that they don't hide any of the body markers).
    - Rest of body is straight.
    - Feet point forward

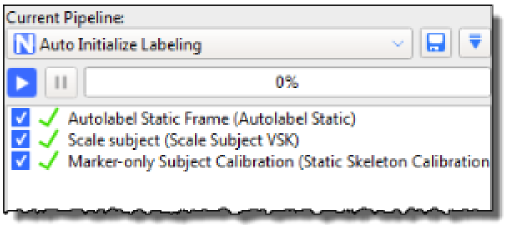


For the rest of the ROM trial, have the subject go through the required range of motion.

> **Tip**
>
> If the subject is not able to attain the base pose, they can use any pose where all of the markers are visible to the cameras. This ensures that the markers can be reconstructed, but is unlikely to produce good autolabeling when **Autolabel Static Frame** (part of the **Auto Initialize Labeling** pipeline) is run. If this is the case, you will need to manually label the trial (see Manually label a trial on page 128) and then run the two remaining operations from the **Auto Initialize Labeling** pipeline. (For information on running pipelines, see Run a pipeline on page 132.)
>
> If the chosen base pose proves unattainable by many subjects, consider substituting a different, attainable pose by specifying a different frame. To do this, on the **Pipeline Tools** pane, expand the **Subject Calibration** section and double-click the **Set Autolabel Pose** pipeline operation to add it to the current pipeline. In the **Properties** pane, ensure the frame with the required pose is selected and then run the **Set Autolabel Pose** pipeline.

13. When enough data has been captured, click **Stop** to end the trial. For a ROM trial, this is typically 1–3 seconds of static data capture, followed by the required ROM. If you are using the **Subject Preparation** tab, Nexus automatically switches to **Offline** mode.

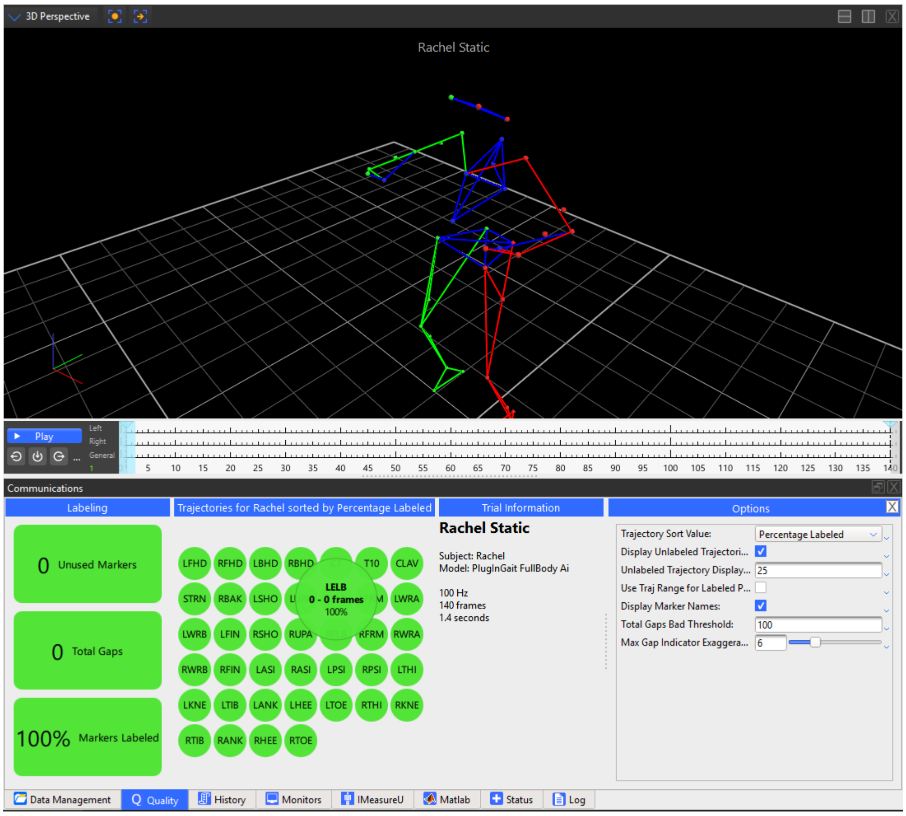
14. Reconstruct the trial either by clicking **Reconstruct** on the Nexus toolbar or, if you need to change the reconstruction settings, by clicking the **Pipeline** button in the **Tools** pane and configuring and running the **Reconstruct** pipeline to create a 3D image of the captured markers. (For information on how to run a pipeline, see Run a pipeline on page 132.)
15. View the subject data in a **3D Perspective** view and ensure that all the markers are clearly visible to the Vicon cameras. (If they are not, check your camera settings, system setup steps, system calibration, etc. Also check that clothing isn't causing markers to move, and the condition (age, etc) of the markers themselves.)
16. From the **Pipeline Tools** pane, run the supplied **Auto Initialize Labeling** pipeline (or if preferred, you can manually label the trial):



17. Nexus labels the trajectories based on the marker set defined in the *.vst* file.
18. Save the trial and the labeling skeleton (VSK) by pressing CTRL+S or by clicking the Save button on the Nexus toolbar. If an asterisk is displayed to the right of the the subject name in the **Subjects Resources** pane, this means that you have also made changes that affect the labeling skeleton template (VST).To save the VST, right-click on the subject name and click **Save Subject**.

Review and finalize a calibration of a labeling skeleton using a ROM trial

1. After you have saved your initial calibration (see above), assess the results by looking at the labeling of the frame that contains the static base pose in the 3D Perspective view. Ensure that all markers are connected by sticks and that the sticks on the left side of the subject are red and those on the right side are green. For a more detailed check, look at the information on the data **Quality** tab.



> **Tip**
>
> To see all the assigned marker labels in the **3D Perspective** view, press CTRL+space bar. This enables you to check that all the required markers are present and correctly labeled.

2. Depending on the results of the labeling, take the appropriate action:

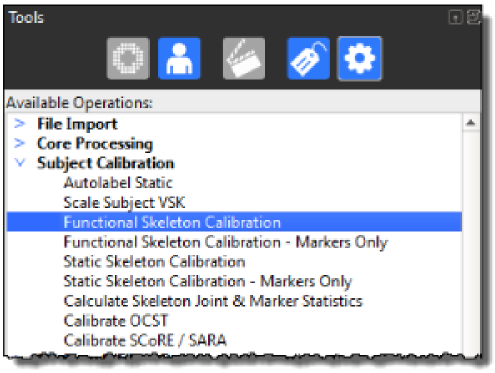
   - If the static frame is correctly labeled and the labeling skeleton fits well, any further processing of your first movement trial (see Capture movement trials on page 136) will give you an indication of the level of labeling that this subject calibration will provide for your specific trial type. Based on these

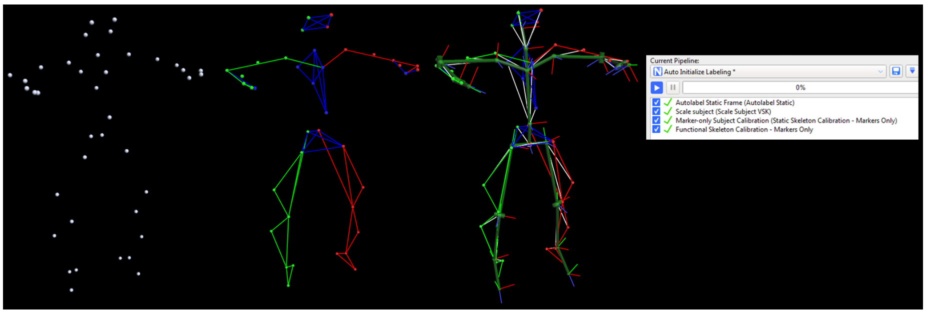
results, you can decide whether they meet your requirements. For example, for walking trials, the quick calibration produced by the **Auto Initialize Labeling** pipeline is likely to produce good labeling of movement trials.
When the subject has been calibrated, segments of the labeling skeleton are fitted to the labeled markers in the **3D Perspective** view, and your subject is automatically labeled. To verify that the subject has been calibrated successfully, switch Nexus to **Live** and ask your subject to move around in the capture volume. You should see that the markers are labeled and the subject is tracked.
If the results are satisfactory, go straight to step 4(set a range).

- If the results are not satisfactory, this may be because your movement trials include more challenging or complex movement and therefore require a further level of labeling reliability. To provide this enhanced reliability you will need to run a functional calibration, which calibrates the subject's bone lengths and joint and marker locations from the whole ROM trial, as described in step 3. To ensure the best results, first consider the following factors:
  - If any markers are labeled incorrectly, fix them (see Correcting swapped labels on page 127 and Manually label a trial on page 128) and from the **Auto Initialize Labeling** pipeline, re-run the **Scale subject** and **Marker-only Subject Calibration** operations; or
  - If all the labels are correct but there is a large distance between a marker and the corresponding reconstruction, the labeling skeleton cannot accurately represent the configuration of labeled reconstructions. This is an indication that you may get poor labeling results and may need to improve your labeling skeleton template (see Requirements for custom labeling skeleton templates in *Creating labeling skeleton templates (VSTs)*).

Note that **Functional Skeleton Calibration** takes more processing time to complete than the **Auto Initialize Labeling** pipeline.

3. (Optional step) Run a **Functional Skeleton Calibration** by completing the following steps:
   a. Verify that the trial is correctly labeled for every frame.
      Note that **Autolabel Static Frame** only labels the trajectories that are present on the static frame(s).
   b. Visually check for any markers that get occluded and manually re-label them (see Manually label a trial on page 128) when they re-appear.
   c. On the **Pipeline Tools** pane, expand the **Subject Calibration** operations section, add the **Functional Skeleton Calibration** operation to the current pipeline and run it.



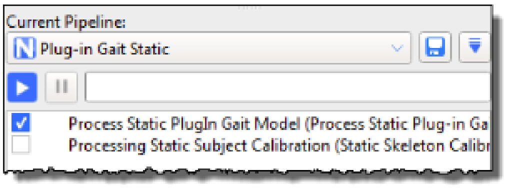


d. When the **Functional Skeleton Calibration** has completed, in the **Options** dialog box (F7), click **Subjects** (on the left) and in the **Properties** pane, go to the **Markers** section, and select **Draw** Covariance.
If you can see any large covariance ellipsoids around a marker, check for mis-labeling of a small number of frames for that marker and fix any mislabels. Note that some markers, such as the ASIS markers, will naturally have a larger covariance, due to skin motion.

> **Tip**
>
> The size of covariance ellipsoids vary, depending upon the way a marker is defined in a skeleton, the amount of marker movement possible, and the quality of the subject calibration. Particularly large covariance ellipsoids may indicate a poor calibration.

4. On the Time Bar, move the **Start Range indicator** and **End Range indicator** (the blue triangles) along the timeline to set a range of frames to select only the first few seconds of the static pose at the beginning of the trial.
5. Run any further processing needed to achieve the required modeling. For example, if you are using Plug-in Gait, to make the necessary calculations and calibrate the static subject, run the **Plug-in Gait Static** pipeline:

a. In the Current Pipeline list, select the Plug-in Gait Static pipeline, and ensure that you leave the check box for Processing Static Subject Calibration cleared (its default setting).



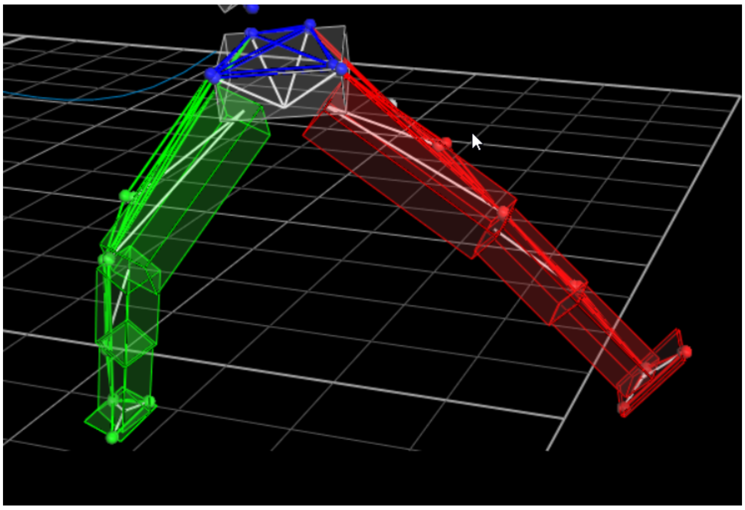
b. Click on the Process Static Plugin Gait Model operation and ensure that in the Properties pane, the First Frame is set to Selected Start and Last Frame is set to Selected End (the default settings).

c. Run the Plug-in Gait Static pipeline.

6. To check that your model has been processed correctly:

- Ensure that in the Pipeline Tools pane, a green check mark is displayed to the left of the operation in the Current Pipeline list.
- Ensure that Plug-in Gait bones are visible in the 3D Perspective view. If they are not, press F7 to open the Options dialog box and select Plug-in Gait Bones.
- In the Subjects Resources pane, ensure that you can expand Model Outputs to see all the calculated components of variables for a kinematic model (such as Angles, Forces, Moments, etc).

  To verify that Nexus has successfully created a labeling skeleton and that it is automatically labeling, ensure that, as shown in the following image, the bounding boxes for Plug-in Gait model segments are displayed in the 3D Perspective view. (The bounding boxes that Nexus draws around each segment that is defined in the template are for visualization purposes only.)

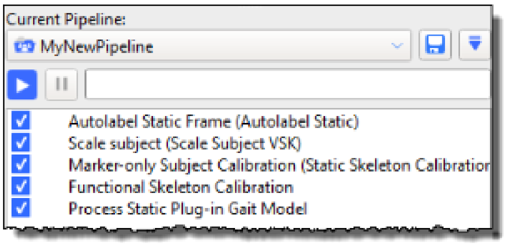


7. Save the ROM trial and the labeling skeleton (VSK) by pressing CTRL+S or clicking the Save button on the Nexus toolbar.

When you have a fully labeled and processed trial (containing only raw reconstructions and no gap-filling), you can proceed to capture and process your movement trials (see Capture movement trials on page 136).

Tip

To save time in future, you may want to save all the relevant pipeline operations to one customized pipeline. To do this, add the relevant operations to the end of the **Auto Initialize Labeling** pipeline and save the pipeline under a new name.



Calibrate a labeling skeleton using a static trial

The recommended workflow for Nexus 2 is to use a ROM trial and dynamic calibration as this offers the benefits of the more efficient, automated labeler (for more information on this workflow, see Calibrate a labeling skeleton using a ROM trial on page 115). However, if you need to use a static frame to calibrate your labeling skeleton (for example, if you are capturing simple movement, or your subject cannot perform a full ROM, and/or you want to obtain some results quickly), you can instead use the following workflow.

To perform a static calibration:

1. Ensure you have obtained and entered any subject measurements required to run the model (required measurements are highlighted in red in the **Subjects Resources** pane): for Plug-in Gait, these are the body weight and height as well as measurements of both left and right leg lengths, knee widths, and ankle widths. These are used to calculate joint center positions, and must be measured and entered into Nexus before any processing can begin. Optionally, you can also measure tibial torsion, the inter-ASIS distance, the ASIS-to-greater trochanter distance, and the sole delta if the subject is wearing shoes.
2. In the **Communications** pane, on the **Data Management** tab, ensure that you have created or opened the required database in which to store the data you will be capturing and that the required session is active. (For a reminder of how to do this, see Prepare a data storage location on page 94.)
3. In the **Resources** pane, ensure Nexus is in Live mode.
4. Display a **3D Perspective** view.
5. Capture a static trial. For more details, see Perform an initial calibration of a labeling skeleton using a ROM trial on page 115.
6. Reconstruct the trial either by clicking **Reconstruct** on the Nexus toolbar or, if you need to configure the reconstruction settings, by clicking the **Pipeline** button  in the **Tools** pane and changing and running the **Reconstruct** pipeline to create a 3D image of the captured markers. (For information on how to run a pipeline, see Run a pipeline on page 132.)

In the **Subjects Resources** tree, right-click on the subject node and attach the required PlugInGait Ai (Auto Initialize) labeling skeleton template (VST) file (see Create a new subject from a template on page 110) to the subject.

> **Tip**
>
> To use a workflow prior to Nexus 2.x, ensure you use the required old labeling skeleton template, (PlugInGait.vst, PlugInGait Fullbody.vst, etc). If you chose to import old settings files when you first installed Nexus 2.x, you can select it from the list of available templates. If you didn't import the old settings files when you installed Nexus 2, copy the old VST files from ...\Vicon\Nexus\ModelTemplates to ...\Vicon\Nexus2.#\ModelTemplates.

7. Ensure that the subject node is the only entry enabled for capture. (When enabled, there is a check mark in the check box.)
8. In the **Pipeline** tools pane, select and run the **Auto Initialize Labeling** pipeline or if necessary, manually label the trial (see Manually label a trial on page 128).
   Nexus labels the trajectories based on the marker set defined in the *.vst* file.
9. Assess the results by looking at the labeling in the **3D Perspective** view. Ensure that all markers are connected by sticks and that the sticks on the left side of the subject are red and those on the right side are green.
10. Save the trial and the labeling skeleton (VSK) by pressing CTRL+S or clicking the Save button on the Nexus toolbar.
11. You can now run any further processing needed to achieve the required modeling and finalize the calibration (see steps 5–7 of Review and finalize a calibration of a labeling skeleton using a ROM trial on page 120).

When you have saved a fully labeled trial (containing only raw reconstructions and no gap-filling), you can proceed to capture and process your movement trials (see Capture movement trials on page 136).

Correcting swapped labels

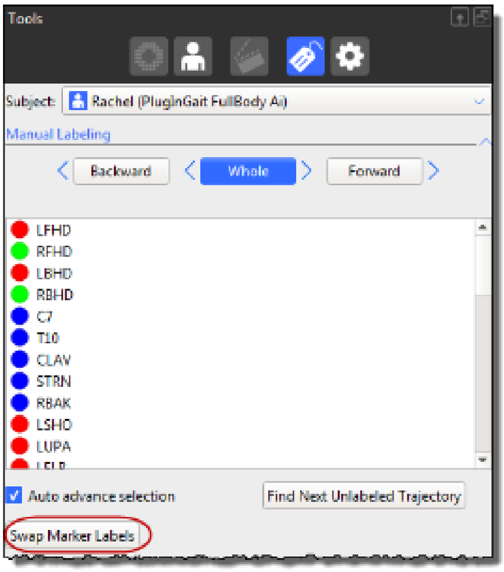
To obtain error-free auto-labeling, it is very important to ensure that the labels are present and correct in your .vst file and that you have labeled your ROM or static trial before running any calibration operation. If the markers in your ROM or static trial are not labeled, the calibration operation will fail. If your markers are erroneously labeled (for example, if left and right markers have been mistakenly swapped) the operation will succeed, but when the Vicon Skeleton (.vsk file) is used for automatic marker labeling of your movement trials, it will continue to produce the erroneous labels.

To address this, check your labeling carefully. If you identify any swapped markers, you can correct this using the **Swap Marker Labels** button in the **Label/Edit Tools** pane.

> **Tip**
>
> To see all the assigned marker labels in the **3D Perspective** view, press CTRL+space bar.

To correct swapped markers:

1. Select the two swapped markers in either the **3D Perspective** view or in the **Subjects Resources** pane.
2. In the **Label/Edit Tools** pane, in the **Manual Labeling** section, click the **Swap Marker Labels** button.



Manually label a trial

The recommended workflow for Nexus 2 is to use a ROM trial and dynamic calibration as this enables the use of automated labeling (for more information on this workflow, see Calibrate a labeling skeleton using a ROM trial on page 115). However, on occasions, you may need to supplement automated labeling with manual labeling, or to use manual labeling where automated labeling gives insufficiently accurate results.

You manually label reconstructed trial data using the tools in the **Label/Edit Tools** pane.

Manual labeling involves associating the markers defined in a Vicon labeling skeleton template (.vst file) with reconstructed markers displayed in the **3D Perspective** view. The manual labeling is then used when the .vst file is scaled to fit the subject wearing the marker set described in the .vst file, during subject calibration. In subsequent movement trials, Nexus uses the resulting subject-specific Vicon labeling skeleton (.vsk file) to automatically label the subject wearing the same marker set.

> ⚠ **Important**
>
> Before manually labeling markers, ensure that you have already:
>
> - Created or opened a session in the database on page 94 in which you want to store the trial data, and ensured that it is the active session.
> - Obtained an existing .vst on page 108 file corresponding to the type of subject whose motion is to be captured, or created a custom Vicon labeling skeleton template.
> - Created a new subject node on page 110 from the template in which the manual labeling is to be stored.
> - Attached the Vicon markers to the subject in accordance with the marker set defined in the associated template. For information on how to do this for Plug-in Gait markers, see Attach Plug-in Gait markers to a patient on page 199.
> - Captured a brief ROM trial including static frames on page 115, or a static trial on page 125, or obtained a single frame of live data with reconstructed markers corresponding to the marker set specified in the associated .vst file.

To manually label 3D marker reconstructions:

1. In the **Communications** pane, on the **Data Management** tab, open the trial file that contains reconstructed markers.
2. On the time bar, move the current time indicator (vertical blue line) to the first frame of the trial in which the subject is standing in a stationary neutral pose. This is typically a T-pose or 'motorbike' pose, in which the subject stands in the basic neutral pose and raises the arms out straight to the sides with palms facing down in a position in the shape of a T. If the subject is not able to attain the T-pose, they can use any neutral pose where all of the markers are visible to the cameras so that they will be able to be reconstructed.
3. In the **Subjects Resources** tree, ensure that the subject node you created from the template is the only entry enabled for capture. (When enabled, there is a check mark in the check box.)
4. At the top of the **Label/Edit Tools** pane, from the **Subject** list select the subject to be manually labeled.
   Labels for the markers defined in the .vst file are displayed in the list in the **Manual Labeling** section.
5. By default, the **Whole** button is selected, so trajectories are labeled in both directions. This ensures that, when you label a marker at the current frame, the same marker is labeled throughout the trial. However, if you want the marker to be labeled from this point forward (or backward) only, for example, if you are resolving an overlapping trajectory, click the **Backward** or **Forward** button before proceeding.
6. In the **Manual Labeling** section, ensure **Auto advance selection** is selected to have Nexus automatically select the next label in the list after you have assigned a label to a marker.
7. In the list in the **Manual Labeling** section, click on the label you want to use.
   The shape of the pointer changes to include a tooltip that identifies the selected label to guide you in assigning labels to markers in the **3D Perspective** view.

   > **Tip**
   >
   > To help you apply labels to the correct markers, on the view pane toolbar, click the **Split vertically** button and in the second pane, open a **Subject Viewer**. Any marker you select in the **Label/Edit Tools** pane is highlighted in the **Subject Viewer**, so that you can see where to place it on the figure in the 3D view.

8. In the **3D Perspective** view click on the marker to which you want to assign the label.
   The next label in the list is selected and the tooltip changes to indicate that label. As you label more markers, a colored line is drawn between each marker until a stick figure appears.

   > **Tip**
   >
   > If you move the mouse to another area of the Nexus workspace while a marker is selected in the **Manual Labeling** section, the cursor retains the tooltip, indicating that Nexus is still in labeling mode. If you need to do something else in Nexus before marker labeling is complete, you can turn off labeling mode in one of these ways:
   >
   > - Press the ESC key.
   > - Click on the **Label/Edit** icon, or any of the other icons in the **Tools** pane toolbar.

9. Repeat steps 7-8 until you have assigned all of the labels to markers.
   The stick figure in the **3D Perspective** view should resemble a skeleton of the subject type defined in the .vst file.
10. To verify that the manual labeling was successful, check that all the markers are connected by sticks and that the sticks on the left side of the subject are red and those on the right side are green.

   > **Tip**
   >
   > To find the next unlabeled marker in the clip, you can also use the **Find Next Unlabeled Trajectory** button. Nexus searches from the current frame forward until it finds an unlabeled marker. When found, the marker is selected and centered in the view pane.

11. To save the labeling information with the trial, on the Nexus toolbar, click the **Save** button.

Work with pipelines

Pipelines enable you run one or more operations that you use frequently or on a large number of trials, from import, through reconstruction and labeling and various other types of processing, to export.

You can set up pipelines to run automatically immediately after capture or as required on one or more saved trials.

You can use any of the pipelines supplied with Vicon Nexus or create your own by grouping a particular sequence of operations to suit your particular needs. Each operation within a pipeline can have a unique set of options and parameters.

> **Tip**
>
> If you are working with large numbers of files, to further speed up your work, you can set up and run pipelines as batch processes, using **Show File Transfer /Batch Processing interface** and then the **Batch Processing** button on the **Data Management tab**.

The following topics describe how to:

- Run a pipeline on page 132
- Create a pipeline on page 134

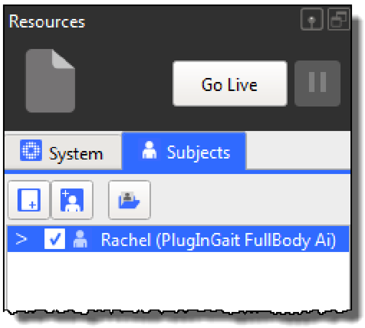
Run a pipeline

This topic describes how to run a pipeline. For information on creating your own pipelines, see Create a pipeline on page 134.

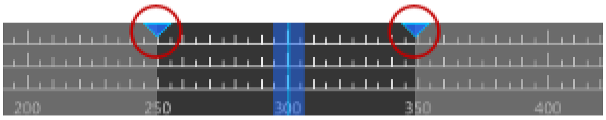
The procedure for running a pipeline is the same, whether you want to run a pipeline supplied with Vicon Nexus or your own custom pipeline.

To run a pipeline:

1. In the **Subjects Resources** pane, ensure that the subject whose trial data you want the pipeline to operate on is enabled. (When a subject is enabled, there is a check mark in the check box next to it.)

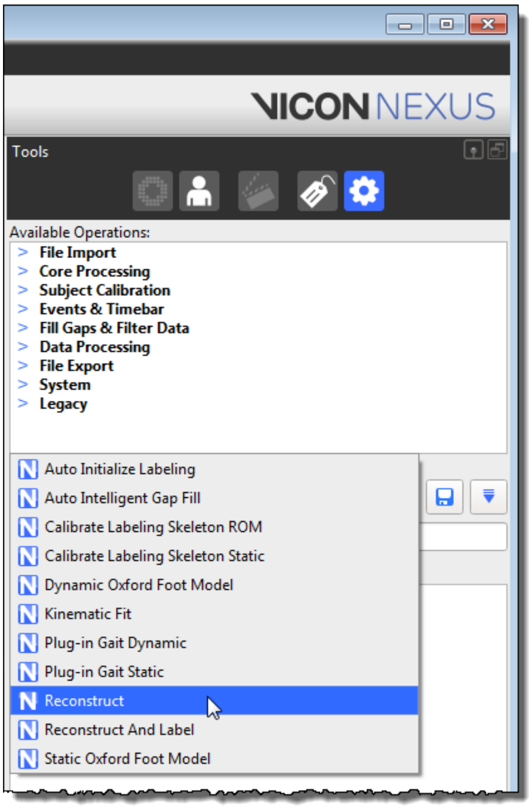


2. If you want the pipeline to operate only on a part of the trial rather than the whole trial, on the time bar, move the blue triangles that represent the start and end of a range of frames along the time line to set the required range.

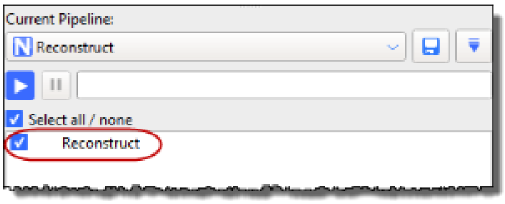


> **Tip**
>
> To change the view of the timeline so that it shows only the selected range, right-click on the timebar and then click **Zoom to Region-of-Interest**.

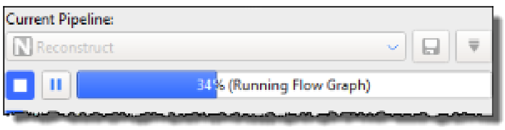
3. In the **Pipeline Tools** pane, from the **Current Pipeline** list, select a pipeline.



4. In the **Current Pipeline** operations list, ensure that the operation(s) to be run is selected.



5. Click the Run button to start the pipeline process. Each operation is run in the order it appears in the list from top to bottom.
   The progress bar displays the results of the pipeline processing, and gives an indication of the percentage of processing completed for each operation.



The **Current Pipeline** operations list may contain the following information for each operation (to the right of the check box):

- **Processing** The operation is being processed.
- **Processed** The operation was completed successfully.
- **Failed** The operation was not completed successfully.
- **Stopped** The operation processing was stopped.

Create a pipeline

To save time, you can save and run sequences of frequently used processing operations using tools in the **Pipeline Tools** pane. Some pipelines can also be selected and run in the **Subject Preparation Tools** pane and the **Capture Tools** pane.

The following instructions describe how to create your own pipeline. For information on how to run an existing pipeline, see Run a pipeline on page 132.

To create a pipeline:

1. If you want to modify an existing pipeline, in the **Pipeline Tools** pane, in the **Current Pipeline** area, select an existing pipeline. If you want to create a new pipeline, go straight to the next step.
2. In the **Available Operations** list, expand or collapse the pipeline type to display or hide the operations you can use for that type of automatic processing of trial data:
   - **File Import** Operations for automating the import of trial files into Vicon Nexus from third-party software packages.
   - **Core Processing** Operations for automating reconstruction, labeling, and kinematic fitting to produce 3D trajectories from raw marker data.
   - **Subject Calibration** Operations for automating the processing of system and labeling skeleton calibration.
   - **Events & Timebar** Operations for automating events and operations that can be carried out using the timebar.
   - **Fill Gaps & Filter Data** Operations for automating the post-processing of data, such as gap-filling and data-filtering.
   - **Data Processing** Operations for automating the production of model outputs (forces and moments, joint angles, etc).
   - **File Export** Operations for automating the export of trial files from Vicon Nexus to third-party software packages.

- **System** Operations for automating offline camera calibration, applying a codec to video, resetting force plate offsets and setting the camera calibration origin.
- **Legacy** Operations for automating access to legacy labeling and fit motion.

For further details of each of the pipeline operations, see Pipeline tools in the *VIcon Nexus Reference Guide*.
To view a tooltip describing an operation's function, hover the pointer over its name in the list.

3. Double-click each operation that you want to include in your pipeline.
The operation is displayed in the list in the **Current Pipeline** section. The operations are run in the order they appear in the pipeline; to rearrange the order, drag operations into the required position in the list.

> **Tip**
>
> To automatically save the trial you are processing, add the **Save Trial - C3D + VSK** operation (located in the **File Export** pipeline operations) to your pipeline. As a general rule, you will want to save the trial as a last step after the other pipeline operations have been run, so position the **Save Trial - C3D + VSK** operation at the required point in the **Current Pipeline** operation list.

4. In the **Current Pipeline** list of operations, click on the operation you added in the previous step, then in the **Properties** pane at the bottom of the **Pipeline Tools** pane, view or change the settings as required.
5. To save your pipeline settings to a *.pipelines* file in the **Pipelines** folder (see Manage system configurations on page 21), click the Save button to the right of the **Current Pipeline** list.
6. In the **Save As** dialog box, supply a name for your new pipeline.
An asterisk * is displayed next to the pipeline name if there are unsaved changes.

Capture movement trials

Before you collect your trial data, you must first have calibrated the labeling skeleton (see Calibrate a labeling skeleton on page 113) and ensured your subject is wearing the appropriate marker set.

When you have calibrated the labeling skeleton, you are ready to capture movement trials for the specified subject, during which the patient performs the motion that is to be analyzed. You then reconstruct and label the trial data, and run any further processing required.

For detailed instructions on these steps, see:

- Capture the required movement on page 137
- Reconstruct and label movement trials on page 150

You capture motion data using the tools in the **Capture Tools** pane, and process trial data using tools on the Nexus menu bar or in the **Pipeline Tools** pane.

Capture the required movement

After you have calibrated the labeling skeleton for your subject (see Calibrate a labeling skeleton on page 113), you can capture the required trials.

You can stream motion data in real time and/or capture raw camera data for offline processing. You can capture a trial manually (see below), or configure Nexus to start /stop capture automatically (see Automatically start and stop capture on page 140).

To quickly capture sequences of trials using the same setup information for each trial, see Use Simple Capture Mode on page 148.

> **Note**
>
> For clarity, reconstruction and labeling are described as separate steps. However, if you want to run reconstruction and labeling as soon as each trial has been captured, you can add these operations (and any others that you want to run automatically) to a post-capture pipeline that runs as soon as capture finishes. For more information see step 7 below.

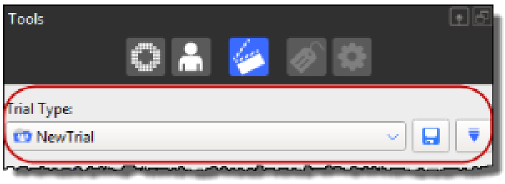
Before capturing a movement trial, ensure that you have already:

- Created or opened a session in the database in which you want to store the trial data, and ensured that it is the active session. For more information, see Prepare a data management hierarchy to store your files on page 94.
- Loaded an existing .vsk file for the subject whose motion is to be captured. If you have just created your VSK (see Calibrate a labeling skeleton on page 113), the correct VSK is already loaded. If not, in the **Subjects Resources** pane, click the **Load an existing subject** button and open the required .vsk file.
- Attached the Vicon markers to the subject in accordance with the marker arrangement used in the labeling skeleton (.vsk) file (if you are using Plug-in Gait, see Attach Plug-in Gait markers to a patient on page 199).

> **Tip**
>
> If you are using Plug-in Gait, Vicon recommends that you do not remove the heel markers for the movement trials. Their presence does not affect the calculated output from Plug-in Gait, but it does improve the reliability of the automatic labeling in Nexus.

To capture movement trials:

1. In the **Resources** pane, ensure Nexus is in **Live** mode.
2. Display a **3D Perspective** view.
3. In the **Capture Tools** pane, select an existing capture configuration for the trial from the **Trial Type** list or save as a new one.



4. In the **Next Trial Setup** section, complete the details for storing your trial data in the active session.
   In the **Data Source Setup** section, ensure **Optical Camera Data**,**Video Camera Data**, and **Device Data** are selected as required.
5. If you want capture to start and/or stop automatically, specify the required settings in the **Auto Capture Setup** section. (For more information, see Automatically start and stop capture on page 140.)
6. If you are conducting clinical gait trials for kinetic analysis involving one or more force plates and want Nexus to automatically assess whether the foot is correctly positioned and is producing valid data for your live trials, ensure that in the **Options** dialog box (F7), **Footstrikes** is selected. For more information, see Automatically assess foot strikes on page 146.
7. In the **Post-Capture Pipeline Setup** section, specify any pipelines containing operations to be run automatically on the trial data when the capture is complete. (For example, if you are using Plug-in Gait, you could select **Run pipeline after capture** and then select the **Reconstruct And Label** and **Plug-in Gait Dynamic** pipelines. This automatically reconstructs and labels the data and then runs filtering, event detection, modeling and export operations.) However, to start with, you will probably prefer to leave this option cleared, so that you can run each operation separately and observe their effects.
8. In the **System Resources** tree, select **Local Vicon System** and then in the **Properties** pane, in the **General** section, set **Processing Output Level** to **Labels**.
9. In the capture volume, have the subject perform the movement that is to be captured.

Nexus automatically reconstructs, labels, and fits the Vicon labeling skeleton created in the previous stage (see Prepare a subject on page 108) to the subject in real time.

> **Tip**
>
> If a labeling error is obvious or persistent, restart the labeler by right-clicking **Local Vicon System** node and then clicking **Reboot Core Processor** (CTRL+R).

If the subject steps on the force plates, the force vector is also shown in real time.

10. On the **Capture Tools** tab, unless you have specified a time or trigger on which to start capturing (see step 5), in the **Capture** section, click **Start** to begin capturing and in the capture volume, have your subject wearing the appropriate marker set for your VST perform the required motion.
11. Unless you have specified a time or trigger on which to stop capturing in the **Auto Capture Setup** section, when the subject has completed the required motion, click **Stop**.
    If you specified a post-capture pipeline (see step 7), Nexus automatically switches to **Offline** mode, displays the subject data in the **3D Perspective** view, and automatically performs any operations contained in any pipelines selected in the **Post-Capture Pipeline Setup** section. Otherwise the system remains Live, ready to capture the next trial.

    > **Tip**
    >
    > If events are not automatically detected, or are only partially detected, you will need to add them manually (see Add events to trials on page 180). You will probably need to do this in situations such as: trials without force plates; movements that are not on a flat surface, such as stepping up/down/over, climbing, etc; pathological or non-cyclical gait movement.

12. Repeat steps 9–11 until you have obtained the required number of trials.

You can now reconstruct and label the trials. For more information, see Reconstruct and label movement trials on page 150.

Automatically start and stop capture

If required, instead of having to manually start and stop capture (see Capture the required movemen on page 137t), you can trigger data capture using an external device or based on a specified period of time.

You can configure Vicon Nexus to automate data capture using the following options:

- Automatically record data before capture is triggered on page 140
- Stop data capture after elapsed time on page 141
- Trigger data capture on labeling percentage on page 141
- Trigger data capture using a remote control device on page 143
- Trigger data capture using timecode on page 143
- Trigger data capture over a network on page 145

Automatically record data before capture is triggered

The **Capture before start** option enables you to specify the number of seconds of data to record prior to capture being triggered either manually (with the **Start** button) or automatically (based on timecode or a remote control device).

To automatically record before capture is triggered:

1. Ensure you have prepared for capture as described in Capture the required movement on page 137, and completed steps 1–4.
2. In the **Capture Tools** pane, expand **Auto Capture Setup**, select **Capture Before Start (secs)** and specify the number of seconds to record data prior to capture being triggered.
3. Trigger the capture manually or using one of the remote capture methods described in this section.
   The Vicon cameras capture the subject moving in the capture volume.
4. Stop the capture manually or using one of the remote capture methods described in this section.
   Nexus automatically switches to Offline mode and displays the subject data in the **3D Perspective** view.

Stop data capture after elapsed time

The **Stop after duration (secs)** option enables you to specify a time period after which Nexus is to automatically stop a capture.

To stop data capture after a specified time period:

1. Ensure you have prepared for capture as described in Capture the required movement on page 137, and completed steps 1–4.
2. In the **Capture Tools** pane, expand **Auto Capture Setup**, select **Stop After Duration (secs)** and specify the number of seconds after which to automatically stop motion capture.
3. Capture a trial manually or using one of the remote capture methods described in this section.
   The Vicon cameras capture the subject moving in the capture volume. Nexus automatically stops the capture after the specified number of seconds, automatically switches to Offline mode, and displays the subject data in the **3D Perspective** view.

Trigger data capture on labeling percentage

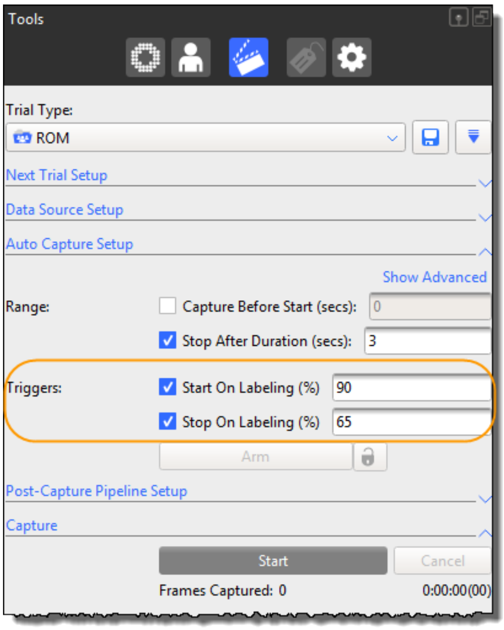
With the **Start on Labeling (%)** option selected, captures automatically start when a subject first fully enters the volume. If the **Stop on Labeling %** option is also selected, captures automatically stop when the subject leaves the volume.

You can set the labeling percentage (ie, the percentage of markers expected from the total number of markers in your subject's labeling skeleton) that needs to be recognized by Nexus within the capture volume for the subject to be considered as fully in the volume or to have left the volume and therefore to trigger capture start /stop.

The following procedure describes how to set up data capture to automatically crop live trials, based on labeling percentages. For information on how to automatically crop trials that you have already saved based on labeling percentages, see Crop trials on page 165.

1. Ensure you have prepared for capture as described in Capture the required movement on page 137, and completed steps 1–4.
2. In the **Capture Tools** pane, expand **Auto Capture Setup** and select **Start on Labeling (%)** and/or **Stop on Labeling (%)**.

3. The default values for this operation start capture when at least 90% of a subject's markers are recognized and stop when less than 65% remain (volume exit). You can adjust these percentages to suit your capture type.



4. Have a subject wearing the required marker set enter the volume.

> **Tip**
>
> To check data quality, you can display the labeling percentage by selecting the subject and in a **Graph** view choosing **Labeling** to show how many labels are present on each frame.

Trigger data capture using a remote control device

The **Start/Stop on remote trigger** option enables you to use an external remote control device to trigger data capture. The remote control device must be connected to a Vicon Lock+ or an MX Giganet in your Vicon system (for details, see either the *Vicon Vantage Reference Guide* or *Go Further with Vicon MX T-Series*), and the sync outputs (GPO pins) for the remote functionality must be configured under the **Vicon Lock** or **MX Giganet** node in the **System Resources** pane.

To trigger data capture using a remote control device:

1. Ensure you have prepared for capture as described in steps 1–4 of Capture the required movement on page 137.
2. In the **Capture Tools** pane, expand **Auto Capture Setup**, ensure the **Advanced** options are displayed, and select **Start/Stop On Remote Trigger**.
3. To set the system to a state where it is ready to accept a trigger signal for automatic capture based on a remote control device, click the **Arm** button.
   If you want to enable the system to remain ready to receive subsequent remote capture signals after the capture is stopped, click the **Lock** button to the right of the **Arm** button.
4. Trigger the start of the capture from your remote control device.
   The Vicon cameras capture the markers on the subject moving in the capture volume and the **Start** button switches to its **Stop** setting.
5. After you have acquired the data you need, trigger the stop of the capture from your remote control device.
   If you clicked the **Lock** button, the **Arm** button is re-enabled and the **Stop** button switches to its **Start** again, ready for a subsequent remote capture.
6. When you have finished your capture session, to review your subject data, manually load the trial.

Trigger data capture using timecode

The **Start On Timecode** and **Stop On Timecode** options enable you to use an external timecode source to trigger data capture in your Vicon system. The timecode source must be connected to a Vicon Lock+ or an MX Giganet in your Vicon system (for details, see either the *Vicon Vantage Reference Guide* or *Go Further with Vicon MX T-Series*), and the corresponding timecode options must be configured under the **Vicon Lock** or **MX Giganet** node in the **System Resources** pane.

1. Ensure you have prepared for capture as described in steps 1–4 of Capture the required movement on page 137.
2. In the **Capture Tools** pane, expand **Auto Capture Setup**, ensure the **Advanced** properties are displayed and select one or both of the following options and specify the required timecode:
   - **Start On Timecode**
   - **Stop On Timecode**
3. Specify the start and stop values in the standard timecode format:
   hh:mm:ss:ff

   where:
   hh = hours (0-23)
   mm = minutes (0-59)
   ss = seconds (0-59)
   ff = frames (0-24 for PAL/SECAM, 0-29 for NTSC)

   For NTSC, the separator character changes between a colon (:) for non-drop frames and a semicolon (;) for drop frames.
   For further details on the use of timecode functionality in Vicon systems, see either the *Vicon Vantage Reference Guide* or *Go Further with Vicon MX T-Series*.
4. To set the system to a state where it is ready to accept a trigger signal for automatic capture based on a timecode, click the **Arm** button.
5. If you want to enable the system to remain ready to receive subsequent timecode signals after the capture is stopped, click the **Lock** button to the right of the **Arm** button.
6. If you selected the **Start On Timecode** check box, start the timecode source from which the data capture is to be triggered; otherwise, under the **Capture** section click the **Start** button.
   When the specified timecode is reached, the Vicon cameras capture the subject moving in the capture volume and the **Start** button switches to its **Stop** setting.
7. If you selected the **Stop On Timecode** check box, when the specified timecode is reached capture stops; otherwise, after you have acquired the data you need, click the **Stop** button.
   Nexus automatically switches to **Offline** mode and displays the subject data in the **3D Perspective** view.

Trigger data capture over a network

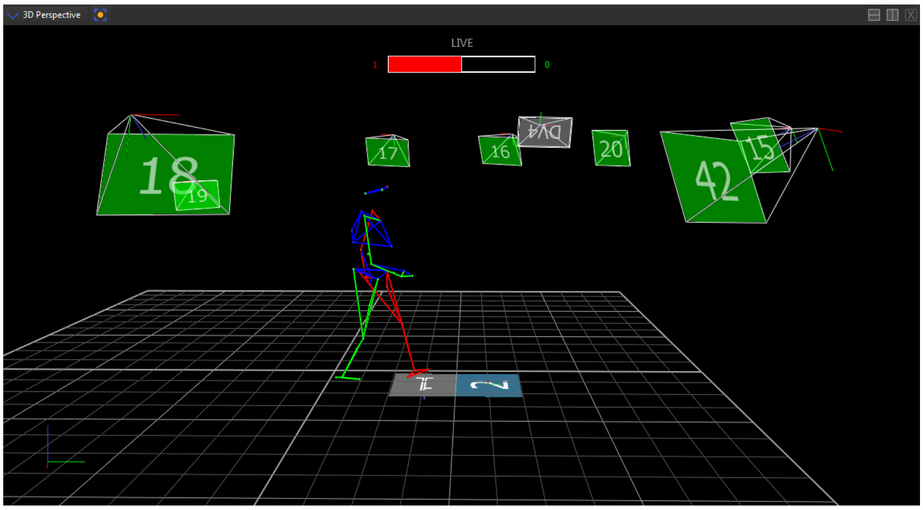
You can broadcast a UDP message over an intranet or direct network cable connection to or from another application (or instance of Nexus) to trigger capture start and stop. Note that Nexus can send or receive the messages, but it cannot send and receive them simultaneously.

To trigger data capture over a network:

1. Ensure you have prepared for capture as described in steps 1–4 of Capture the required movement on page 137.
2. In the **Capture Tools** pane, expand **Auto Capture Setup**, ensure the **Advanced** options are displayed, select **Start/Stop Over Network** and from the adjacent drop-down list, select **Send** or **Receive**.
3. In the **Address** field, either select the IP address of the network card that will be used to send or receive the start/stop trigger message, or select **All**. In the adjacent field, specify the UDP port which is to send or receive the message.
4. To set the system to a state where it is ready to accept a trigger signal for automatic capture based on a signal broadcast over the network, click the **Arm** button.
5. If you want to enable the system to remain ready to receive subsequent network signals after the capture is stopped, click the **Lock** button to the right of the **Arm** button.
6. Start the capture from the remote software. You can start the capture using a timecode or immediately.

Automatically assess foot strikes

In clinical gait trials for kinetic analysis involving one or more force plates, Vicon Nexus can automatically provide an indication of whether or not the foot is correctly positioned and is producing valid data for your live trials. Nexus displays this information on screen. This can save you time when you are assessing each foot strike to decide whether it is valid.



As each strike is detected, or is determined to be invalid, the box representing the force plate in the **3D Perspective** view turns the appropriate color:

- Red : left foot strike
- Green : right foot strike
- Dark gray : invalid strike

Counters on either side of the force plate box show how many valid strikes have been detected for the left foot and the right foot during the current session.

To use auto-detection of foot strikes:

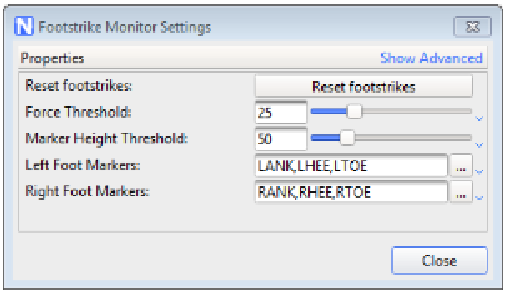
1. Before you begin, ensure that you have:
   - Calibrated the Vicon system.
   - Applied the appropriate marker set to the subject.
   - Ensured that you have a valid subject in Nexus.
   - Connected and configured one or more force plates.
2. On the **Windows** menu (or in the **3D Perspective** view, right-click to access the context menu), click **Footstrike Monitor Settings** and in the dialog box, ensure the options for foot strikes are as required:

a. **Reset footstrikes** button enables you to reset the foot strike counters.

b. **Force Threshold**: Minimum force required on force plate to produce foot strike

c. **Marker Height Threshold**: Minimum height of foot marker above force place for foot to be recognized for foot strike

d. **Left** and **Right Foot Markers**: Comma-separated list of names of one or more markers that define the segment (i.e. foot) that will strike the plate.



3. In the **Options** dialog box (press F7), ensure that **Footstrikes** is selected and that the display options for **Font Size**, **Opacity**, and **Background Color** are as required.

To reset foot strike counters:

- Right-click anywhere in the workspace and then click **Reset**.

or

- Press CTRL+SHIFT+R

Using multiple foot plates

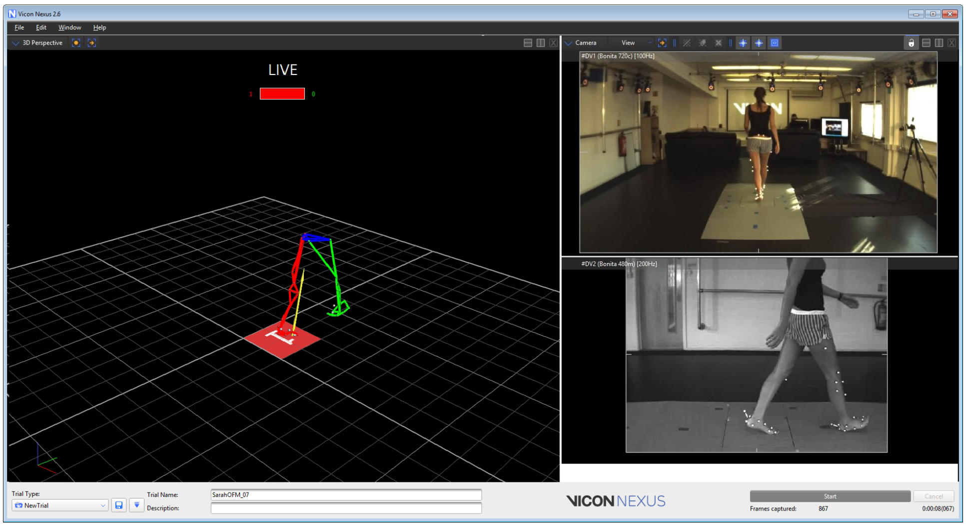
If your trial requires multiple force plates, they are laid out in the order of the force plate IDs (that is, FP1 is furthest left/uppermost depending on the position of the force plate box).

Strikes straddling two force plates are marked invalid. However, if required they can be processed by the **Process Dynamic Plug-in Gait Model** pipeline operation, by selecting the **Allow cross-plate strikes** option.

Use Simple Capture Mode

Simple Capture Mode enables you to use a workspace that maximizes your view of the capture volume and exposes a focused subset of controls required for repeated captures, allowing you to concentrate on the movement you are capturing.

In **Simple Capture Mode**, only the currently selected view type is displayed, together with controls that enable you to specify the trial type, name, and description, and to start and stop (or cancel) captures.



To display a simplified capture view:

1. Ensure that you have set up your Vicon system as required, that Nexus is in Live mode, and that you have set up your trial types, including any post-capture pipeline required, auto-start triggers, etc. (For information on setting up trials, including pipelines and triggers, see Capture the required movement on page 137 and Automatically start and stop capture on page 140.)
   By default, Simple Capture Mode displays the same view as the current workspace. You can customize the view to include other panes (eg, a video pane alongside the **3D Perspective**), as described in the following steps.
2. Associate a custom view type with **Simple Capture Mode**. To do this:
   a. Open the **Options** dialog (press F7) and on the left click **Simple Capture Mode View Options**.
   b. On the right, from the **View Type** list, select **On**.
   c. From the **User Specified Simple Capture Mode View** list, select one of the following:

- To use the Vicon-supplied **Simple Capture** view type (which displays the view pane with a **3D Perspective** view on the left, and **Camera** views with video cameras selected on the right, as shown above), ensure **Simple Capture** is selected; or
- To use your own view type (for a reminder of the controls that enable you to define and save a view type, see Get to know Vicon Nexus on page 21), select the required option.

3. Turn on Simple Capture Mode. To do this:
   - On the **Window** menu, select **Simple Capture Mode**; or
   - Press CTRL+H

   The simplified capture view that you selected in the **Options** dialog box is displayed.
4. To capture trials, at the bottom right of the Simple Capture view, click **Start**.
5. To exit Simple Capture Mode, either press Esc, or press CTRL+H again. (You can also click the **Window** menu and clear **Simple Capture Mode**.)

Reconstruct and label movement trials

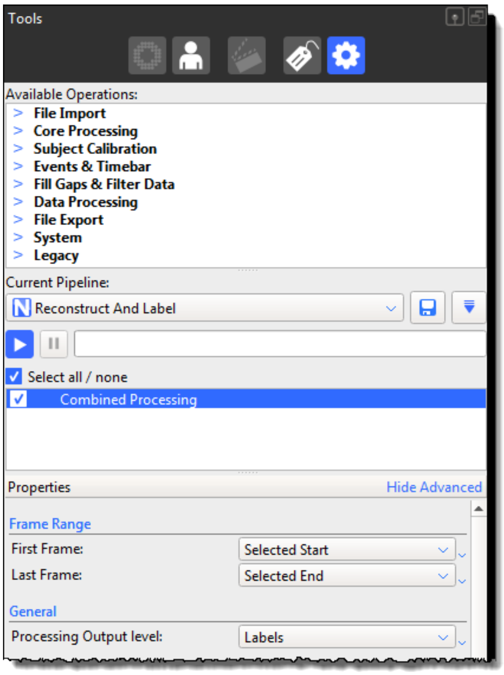
After you have calibrated a labeling skeleton for your subject (see Prepare a subject on page 108) and captured the required movement of the subject in a trial or series of trials (see Capture movement trials on page 136), you must reconstruct and label the raw camera data to create 3D markers. You can do this as a separate step, as described below, or you can specify a pipeline that includes **Reconstruct** and **Label** in the **Post-Capture Pipeline Setup** (see Capture the required movement on page 137), so that these operations run straight after capture.

> **Note**
>
> Note that you can choose to reconstruct and then label separately, or run both operations together, by clicking either the **Reconstruct** button or the **Reconstruct and Label** button on the Nexus toolbar, or by configuring and running the equivalent pipeline operations. Normally, you would use **Reconstruct** alone if you are working on a labeling skeleton template (VST), but **Reconstruct and Label** if you are processing movement trials.

To reconstruct and label trial data:

1. Display the subject whose data is to be reconstructed in either of the following ways:
   - If you have just captured a trial, ensure that the raw marker data for the subject in the capture volume is still visible in a **Camera** view and in the **Resources** pane, ensure Nexus is in **Offline** or **Pause** mode; or
   - On the **Data Management** tab, open the desired trial file containing raw trial data for the subject.
2. Do one of the following, depending on whether you need to adjust the reconstruct and labeling settings:
   - To run the operations without viewing and changing their settings, on the Nexus toolbar, click the **Reconstruct and Label** button.
     This runs the **Reconstruct and Label** operation defined in the **Pipeline Tools** pane.
     Or

- To view and/or change the operations, on the **Pipeline Tools** pane, from the **Current Pipeline** list, select **Reconstruct And Label**. In the list of current operations, click on **Combined Processing** to select it. set the **Processing Output level** to the required level, for example, **Labels**, and specify any other required properties.



The pipeline progress bar indicates when the operation is complete.

You can now review the results and fill any gaps in the trial data. For information, see Review trials and fill gaps on page 152.

Review trials and fill gaps

When you have reconstructed and labeled your movement trials (see Capture movement trials on page 136), you can review the results and fill any gaps in the trial data.

1. If other Nexus users may have worked on your trial data, you may first want to check on any processing that has been performed on the files (see Review processing history on page 153).
2. Review data quality by playing through the trial using the **Time Bar** and/or looking at the information on the data **Quality** tab (see Review data quality on page 154).
3. If necessary, crop any large gaps at the beginning and/or end of the trial, either manually or automatically (see Crop trials on page 165).
4. Fill any gaps using either automatic or manual gap-filling (see Fill gaps in trial data on page 169).
5. If required, you can add events to trials (see Add events to trials on page 180).

After you have filled any gaps in your trial data, you can run the required modeling operations. For more information on modeling, see Modeling with Plug-in Gait on page 185.

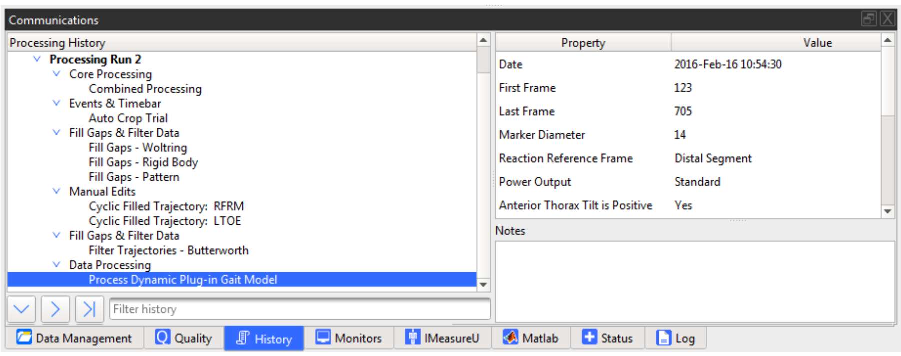
Review processing history

You can review all processing that has been performed on a loaded trial file on the **History** tab in the **Communications** pane. This ensures that, even if you have not worked with the data before, you can work on the trial without missing or duplicating processing steps. For example, if you work as part of group, when you open a trial that someone else has worked on, you can immediately see what processing has occurred (eg, filtering, gap-filling) and what settings were used.

Being able to view processing history helps you to understand your data, prevents errors introduced by running certain operations more than once (eg, filtering), and reduces the need to reprocess data because its current state is not well known.

To view processing history:

1. Open a trial.
2. In the **Communications** pane, click the **History** tab.
   A summary of the data processes and major events is displayed.
3. To display information about a particular event, click on the relevant event. Details about the settings used to run the event are displayed in the **Property** and **Value** columns on the right.



In the **Notes** field, you can add notes that will be useful to you or anyone else working with the same trial. Your notes are saved with the trial.

> **Note**
>
> Processing history is available only for files processed in Nexus 2.3 and above.

Review data quality

To enable you to review the data you captured in your movement trials (see Capture movement trials on page 136), Vicon Nexus offers a number of tools. You can use the time bar to manually scrub through the trial, or to assess your trials quickly and easily, you can use the automated data quality features of Nexus.

For more information, see:

- Review trial data using the time bar on page 154
- Assess trial health with the data Quality pane on page 156
- Detect gaps in the Graph view on page 158
- View gaps with the Data Correction view on page 159
- View data quality across a trial on page 161
- Navigate to gaps using the data quality features on page 163

Review trial data using the time bar

To review trial data using the time bar:

1. Display reconstructed and labeled markers in either of the following ways:
   - Ensure that a trial you have just captured has been reconstructed and labeled.
   - From the **Data Management** tab, open a previously captured and labeled trial.
2. In the **Resources** pane, ensure Nexus is in **Offline** mode.
3. Display a **3D Perspective** view and optionally a **Graph** view.

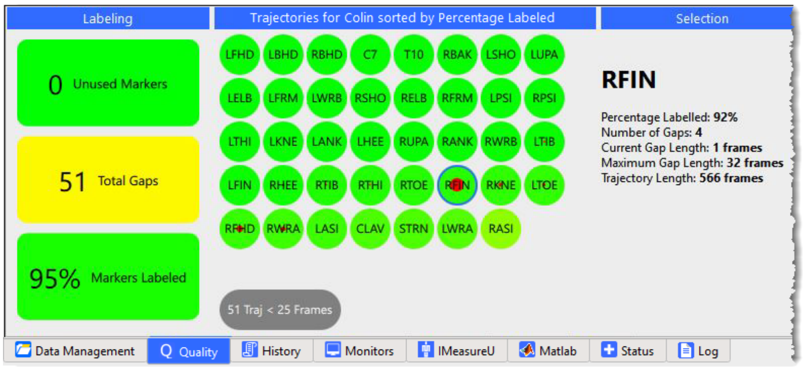
> **Tip**
>
> To automatically display both views, in the **View Type** list on the Nexus toolbar, select **Data Correction**. Using a split layout with both a 3D Perspective and a Graph view enables you to display the results of your editing actions and to select specific trajectories, frame ranges, and gaps in trajectories. Since the graphs indicate what changes have been made to the data and by which tool, you can immediately see the results of any edits you have made and determine if you want to change it again or adjust the data, using a different tool in the **Gap Filling** section.

4. Play back the desired portion of the trial:
   - **Full trial**: In the controls at the left end of the time bar, click Play to start playing the trial data. The data plays through to the end and restarts at the beginning if you do not click Stop.
   - **Range of frames**: Do one of the following:
     - To manually adjust the range, move the blue triangles that indicate the start and end of the selected range of frames (also known as the region of interest) along the timeline; or
     - On the time bar, right-click and then click **Set Region-of-Interest**. In the **Set Region of Interest** dialog box, specify the **Start Frame** and **End Frame**.
   - **Individual frame**: Move the blue vertical line that indicates the currently selected time along the timeline to manually locate the desired frame, or right-click on the time bar, click **Set Current Frame** and enter the number of the required frame.

> **Tip**
>
> To display a context menu in which you can reset the timescale on the timeline, set the playback speed, or specify the length of trajectory tails, click the ellipsis (...) in the time bar controls, or right-click the time bar.

Assess trial health with the data Quality pane

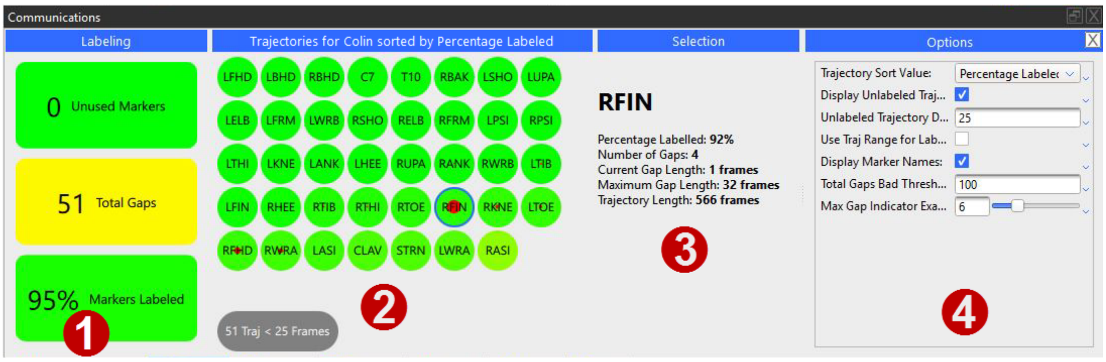
The data **Quality** pane, a tab in the **Communications** window, provides both a broad overview of data quality for a subject across the entire trial and a way to examine the health of individual trajectories. It also enables you to navigate to the highlighted issues.



The three most common data issues that you need to find and resolve are:

- Missing markers (usually knocked off before capture)
- Unlabeled or incompletely labeled trajectories
- Gaps in trajectories

The **Quality** tab helps you to find this information:



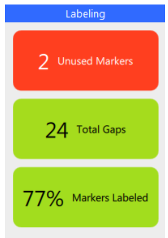
1 **Overview of the health of the whole trial (Labeling column)** The text in each rectangle indicates the number of unused markers, any gaps, and the percentage of markers labeled. As a quick indicator, to enable you to assess whether to continue working with the trial, the color green indicates good health, red indicates probable major issues, and colors between the two (yellow, amber, orange) indicates that the overall health of the trial lies somewhere between these extremes.

For example, a good trial, worth further work, would be indicated by:

- No missing markers
- A high percentage of labeled markers
- Only a small number of gaps.

A low quality trial, which would probably have to be re-run, would be indicated by:

- Missing markers, and/or
- A high number of gaps, and/or
- A lower percentage of labeled markers.



In this example, two required markers are completely absent from the trial, so, without further work, you would know that you would have to either re-reconstruct or (more likely) re-capture the trial.

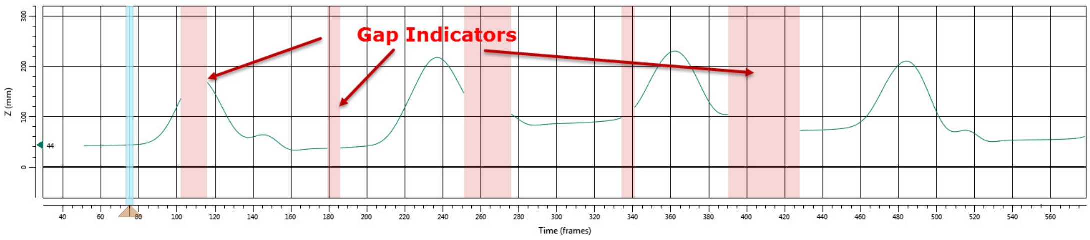
2 **Individual trajectories.** If you hover the mouse over a blob, the blob displays information about the trajectory. If you click on it to select it, information about the trajectory is displayed in the **Selection** column on the right. For more information, see Navigate to gaps using the data quality features on page 163.

3 **General trial information/selected trajectory Information (Selection column).** If no blob is selected, general information about the currently loaded trial is displayed. If a blob is selected, information about the trajectory is displayed.

4 **Display options (Options column):** Enables you to configure the display options for the data **Quality** tab. Alternatively, you can open the **Options** dialog box (F7), click **Data Quality Panel Options** on the left, and change the properties as required. To save any changes to these settings, click the **Save current configuration** button in the **Options** dialog box.

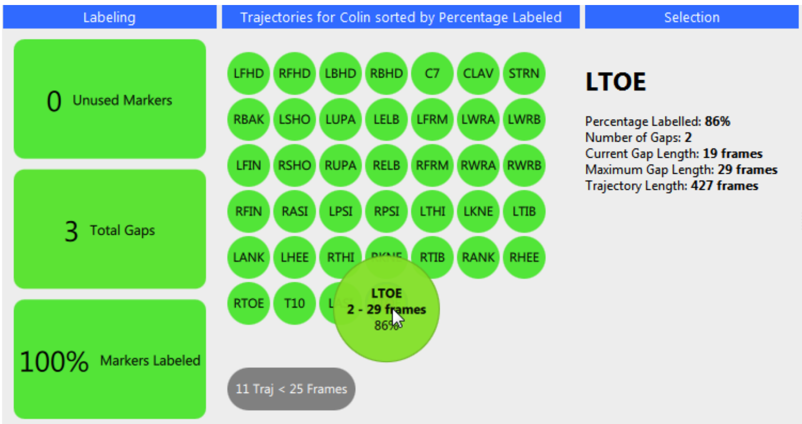
Detect gaps in the Graph view

The **Graph** view enables you to view and fix gaps for a selected trajectory.

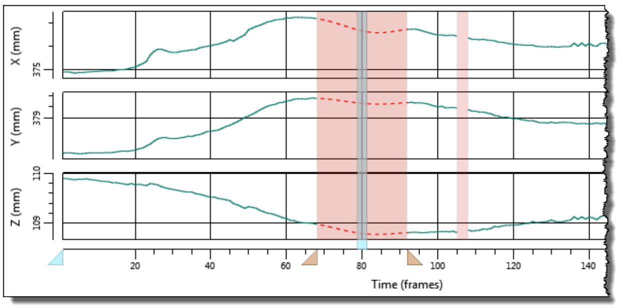


To display gaps:

1. On the **Quality** pane, click a trajectory blob that is reporting a gap.



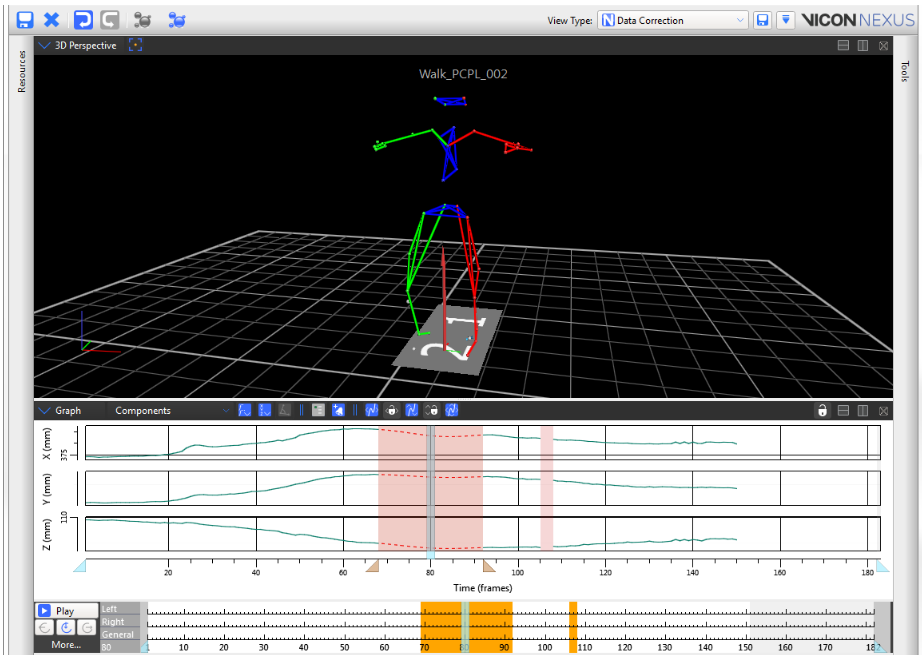
Gap indicators are displayed in pink in the **Graph** view.



2. To zoom in and out, CTRL+double-click on a gap indicator (the pink area) on the graph.

View gaps with the Data Correction view

The default **Data Correction** view type displays a **3D Perspective** view above a **Graph** view.



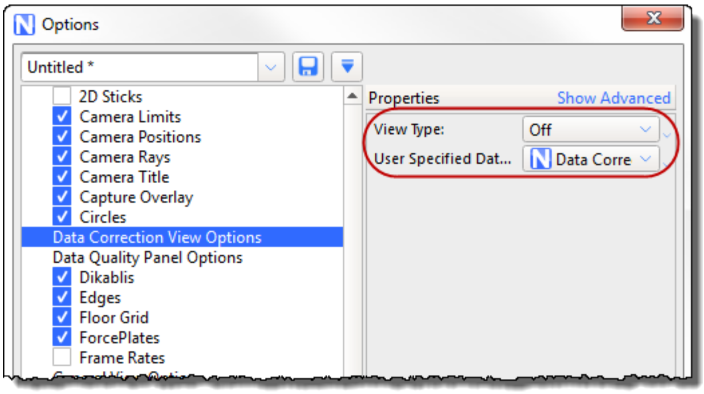
You can turn **Data Correction** view on or off in the **Options** dialog box, as described below.

When the **Data Correction** view is turned on, it is automatically displayed when you navigate using the data **Quality** pane.

If you would prefer the **Data Correction** view to appear differently (for example, displaying the **Graph** above the **3D Perspective** view), you can save your own preferred view and select this instead.

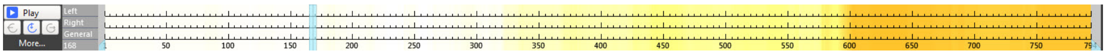
To configure the **Data Correction** view:

1. Open the **Options** dialog box (F7).
2. Click on **Data Correction View Options** on the left and change the properties as required on the right:
   a. If you want to display this view type automatically when you are navigating via the **Quality** tab, ensure **View Type** is set to **On**.
   b. If you have saved a different view type to be displayed when you are using the **Quality** tab, ensure it is selected in the **User Specified Data Correction View** field.



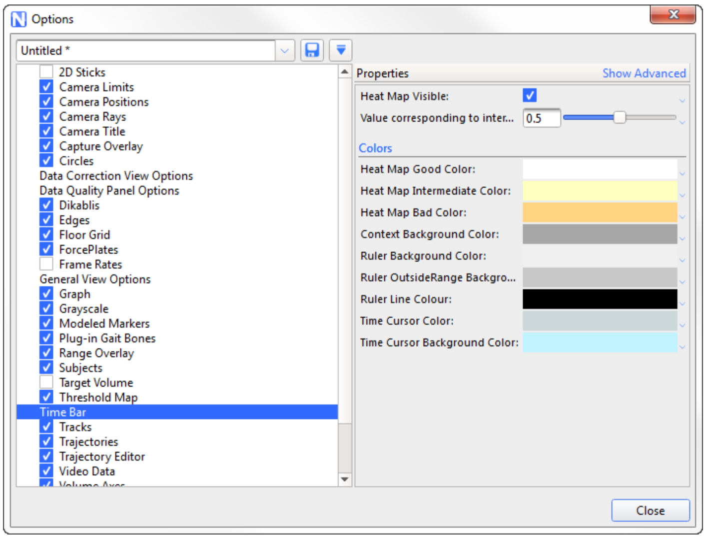
View data quality across a trial

In addition to viewing data quality on the Quality tab and Graph and Data Correction views, you can also view data quality as a heat map on the time bar, to get an overall picture of the current trial.

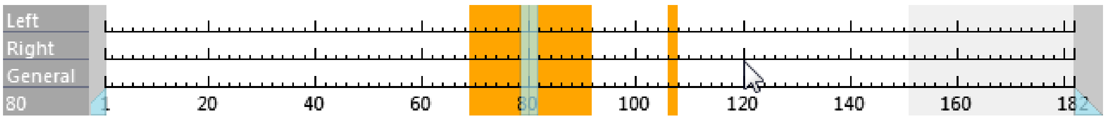


To turn on the heat map overlay:

1. In the **Options** dialog box (F7), click on **Time Bar** on the left, and select the **Heat Map Visible** check box on the right.
2. If required, change the colors of the heat map.



When you click on a trajectory on the **Quality** tab, the time bar shows gaps for the selected trajectory.



Viewing data quality for multiple trajectories

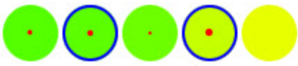
To select multiple trajectories, on the **Quality** tab, CTRL+click the required blobs.

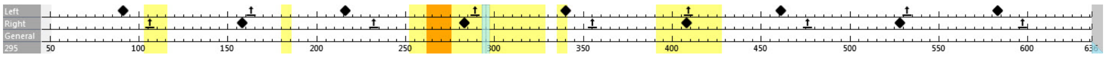
When multiple trajectories are selected, the heat map's color gradient system is activated.

Lighter colors show where one of the trajectories has a gap but other selected trajectories do not. Darker areas show where a number of selected trajectories has a gap.

Two trajectories selected

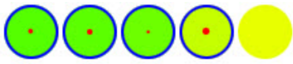
The following images show two trajectories selected. The darker area indicates gaps in both trajectories.

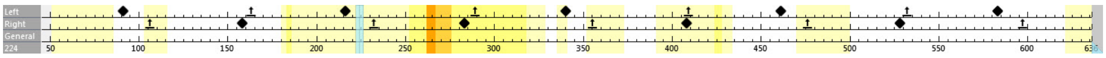




More trajectories selected

As more trajectories are added the view starts to morph from individuals gaps for one or two trajectories, to areas in the volume/time where there are issues.





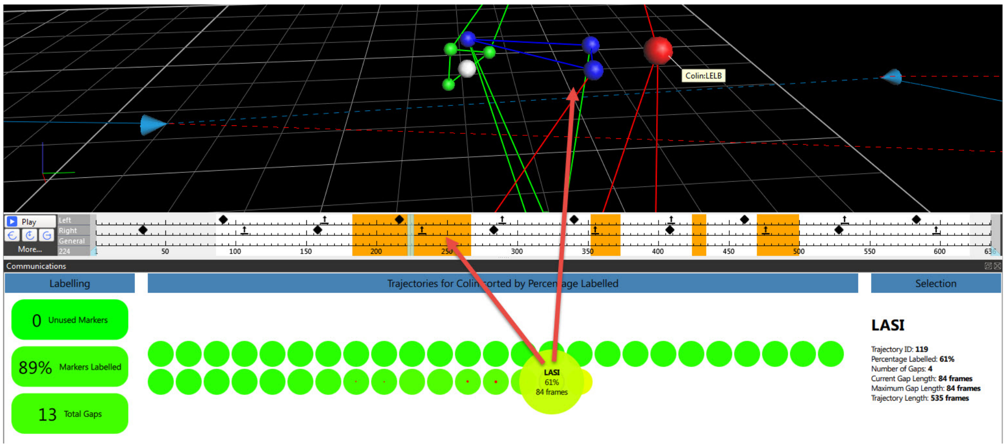
Navigate to gaps using the data quality features

Using the **Quality** tab and the **Data Correction** view type, you can quickly navigate to gaps in data, and use the **Zoom**, **Translate**, and **Rotate** options for a better view.

Quick navigation to gaps

To quickly navigate to gaps:

1. On the **Quality** tab, click on a trajectory blob.
   Immediately, the time bar and view pane displays the first gap (if any) for the selected trajectory.



2. To move to the next gap or previous gap for the selected trajectory, press the hot keys CTRL+8 and CTRL+7 respectively.

Tip

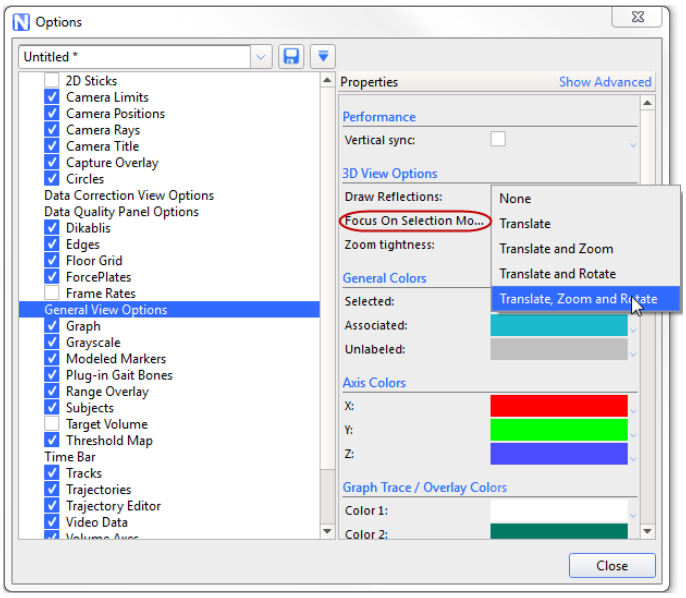
On the data **Quality** tab, you can move to the next trajectory and previous trajectory by pressing CTRL+PgUp and CTRL+PgDown.

Get a better view of gaps

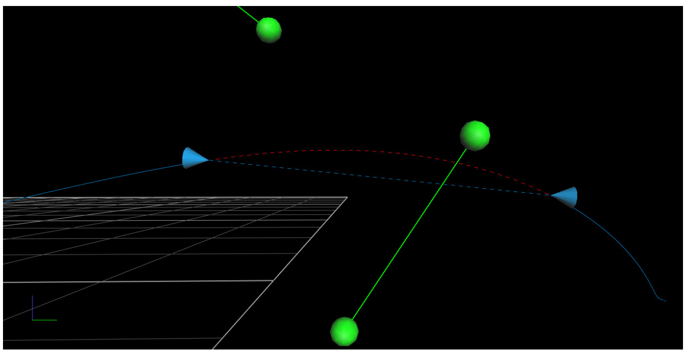
The Zoom, Translate and Rotate options for viewing a gap make it easier to zoom in on and fix selected gaps.

To view gaps more clearly:

1. In the **Options** dialog box (F7), click **General View Options** on the left.
2. In the **Properties** pane on the right, in the **3D View Options** section, click the **Focus on Selection Mode** list.
3. To display a zoomed view that rotates the workspace to a flat perspective of the gap, click **Zoom, Translate and Rotate**.



4. On the **Quality** tab, click the blob for the trajectory whose gaps you want to examine.
   The **3D Perspective** view zooms in and rotates as necessary to clearly show the selected trajectory's first gap.



Crop trials

For trials where the subject starts outside the volume, moves into the volume and then exits, cropping the start and end of the trial removes unwanted frames where the subject was not fully in the volume.

Before you fill gaps in your trial data, it's a good idea to remove these start and end frames as they are likely to contain large gaps and other invalid data. You can do this automatically while capturing (see Trigger data capture on labeling percentage on page 141), or for trials that have already been saved, you can crop trials manually or run a pipeline operation to do this automatically.

Cropping a trial affects only the .c3d file. The raw files remain unchanged.

> **Tip**
>
> If you add gait cycle events and then crop a trial, you will probably also want to delete events that are outside the cropped region. For information on how to do this, see Delete events that are outside the region of interest on page 183 .

For more information, see:

- Manually crop a trial on page 166
- Automatically crop a trial on page 167

Manually crop a trial

You can use tools on the time bar to remove unwanted frames at the start and end of your trials.

To manually crop a trial:

1. Ensure the required trial is reconstructed and labeled (see Reconstruct and label movement trials on page 150).
2. On the time bar, do one of the following:
   - Move the range indicators (blue triangles) to exclude the unwanted frames at the beginning and/or end of the trial; or
   - Right-click and then click **Set Region-of-Interest**. In the **Set Region of Interest** dialog box, specify the **Start Frame** and **End Frame**.

   The gray shaded area represents the cropped frames of the trial.
3. Right-click on the time bar and click **Zoom to Region-of-Interest**.

You can now fill any remaining gaps.

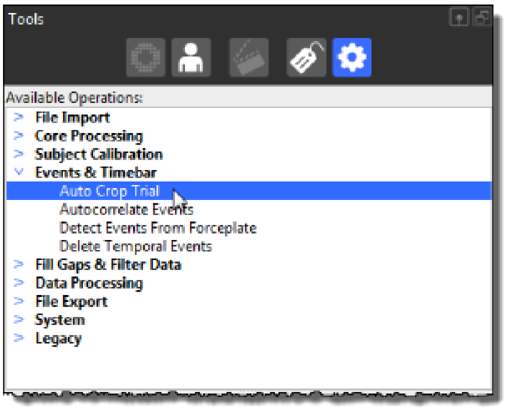
Automatically crop a trial

Nexus can save you time and effort by automatically determining the first and last frame where the subject is fully in the volume

When you run the autocrop operation, Nexus automatically zooms the time region of interest to the points where the subject first fully enters and then leaves the volume.

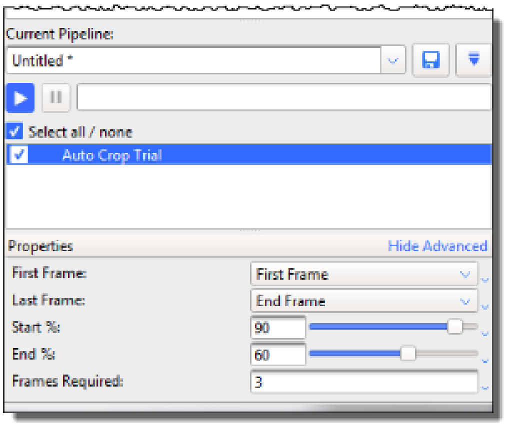
To set up auto-cropping:

1. Ensure the required trial is reconstructed and labeled (see Reconstruct and label movement trials on page 150).
2. In the **Tools** pane, click the **Pipelines** tab and in the **Available Operations** list, expand **Events & Timebar**.
3. Double-click the **Auto Crop Trial** pipeline operation to add it to the current pipeline.



4. In the **Current Pipeline** list, click **Auto Crop Trial** and in the **Properties** pane, ensure that the **First** and **Last Frame** options are set to the beginning and end frames during which the subject enters and leaves the volume.

5. Set the **Start** and **End (%)** options to the required minimum percentage of markers (ie, the percentage of markers expected from the total number of markers in your subject's labeling skeleton) that must be labeled in each case.



6. If necessary, adjust the **Frames Required** to set a minimum number of frames where the **Start %** criterion must be met. This is helpful if early single frames of data meet the %, but the point at which the trial is intended to start (ie, the point at which the % is maintained over a number of frames) is later in the trial.
7. Run the **Auto Crop Trial** operation.
   The trial is automatically cropped to the frames you have selected.

You can now fill any remaining gaps.

Fill gaps in trial data

When Nexus reconstructs each marker on a subject, ideally it produces a smooth trajectory throughout the trial. Realistically, there may be frames with breaks in trajectories due to some markers that could not be reconstructed or spurious data.

> **Tip**
>
> **Auto Gap Fill** attempts to fill all gaps in the trial data. For this operation to work well, you are likely to need to change the default property settings, including the gap size. For data containing large gaps, you are advised to review and manually fill the large gaps first, using the tools in the **Label/Edit Tools** pane, and/or cropping out any unwanted frames with large gaps that occur at the start or end of the trial.
>
> For information on cropping frames at the start and end of the trial data, see Crop trials on page 165.

For more information, see:

- Manually fill gaps in trial data on page 170
- Automatically fill gaps in trial data on page 177
- Example of creating a custom pipeline for gap-filling on page 179

In addition, a Vicon Nexus 2 tutorial video, showing gap-filling operations, including many additional tips and examples, is available on YouTube.

> **Note**
>
> As the videos were recorded using an earlier version of Nexus 2, you may notice small differences in the user interface.

For additional tips on issues you may encounter when using manual labeling, see Improve manual labeling in the *Vicon Nexus Reference Guide*.

Manually fill gaps in trial data

You can manually fill any gaps in the reconstructed and labeled trial data by using the tools in the **Label/Edit Tools** pane.

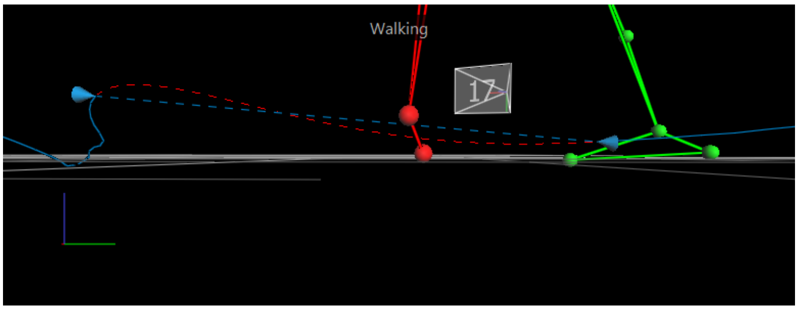
As well as automatically filling gaps (see Automatically fill gaps in trials on page 177), you can manually fill gaps to correct any errors or inconsistencies in the reconstructed and labeled trial data.

To manually fill gaps in reconstructed and labeled data:

1. Display reconstructed and labeled markers, ensure Nexus is in **Offline** mode, and display a **3D Perspective** view and optionally a **Graph** view, as described in Review trial data using the time bar on page 154.
2. Unless you have already cropped your trial (see Crop trials on page 165), to simplify cleaning up your data, set the region of interest of the trial that you wish to analyze. For example, if the capture includes the subject entering and leaving the capture volume, Vicon recommends that you set the range of frames to exclude these parts of the capture, as they are likely to include large gaps. To do this, on the time bar, move the blue range indicator triangles to select a range of frames and then right-click and click **Zoom to Region-of-Interest**.
3. In the **Subjects** tree, ensure that the desired subject is selected (when selected, there is a check mark in the check box).
   In the **Label/Edit** tools pane, in the **Gap Filling** section, any markers whose trajectories contain gaps within the selected range of frames are listed in the **Trajectory** column, with the number of gaps for each trajectory identified in the **#Gaps** column and the largest gap length in the **Max Gap Length** column.

   > **Tip**
   >
   > To jump to the largest gap for a particular trajectory, double-click in its **Max Gap Length** column. To change the sort order of the columns, click the column headings.

4. In the **Trajectory** column, click on the trajectory whose gaps you want to fill. Nexus automatically selects the gap by placing the current time in the middle of the gap and positioning the gap range selectors (blue cones) at the start and end of the gap. A red dotted line is displayed to preview the shape of the trajectory if a spline fill editing operation is run. If you can't see a red dotted line, in the **Label /Edit Tools** pane, in the **Gap Filling** section, hover the mouse pointer over the **Fill** button in the **Spline Fill** area to display a tooltip.



5. In the **Range** section, view the range values to identify the size of the gap and use the buttons to navigate between the gaps in the selected trajectory:
   - **Prev Gap**: Navigate to the previous gap in the selected trajectory. This button is available only if there are gaps in the trajectory before this point.
     - Left edit range: The frame before the start gap range selector, indicating the start of the gap that will be filled in.
     - Right edit range: The frame after the end gap range selector, indicating the end of the gap that will be filled in.
   - **Next Gap**: Navigate to the next gap in the selected trajectory. This button is available only if there are gaps in the trajectory after this point.
6. If required (for example, if the data leading up to the gap is noisy), to extend the edit range beyond the gap, in the **3D Perspective** view, drag the gap range selectors (blue cones).

   > If you need to adjust the size of the cones for easier viewing or selection, open the **Options** dialog box (press F7) and click **Trajectory Editor**. Adjust the value of **Radius** as required (the default is 10 pixels).

7. Use the appropriate fill tool to generate data to fill the selected gap.

   > **Tip**
   >
   > If your chosen tool is unavailable, hover the mouse pointer over the unavailable button to display a tooltip that explains how to use it.

   The fill options are:

- **Spline Fill**: Performs a cubic spline interpolation operation to fill the currently selected gaps. Use it when you have suitable frames with no gaps on either side of the gap. If there are gaps in these frames, the fill is rejected and you will need to choose a different gap-filling method. For more information, see Fill gaps with Spline Fill on page 173.
- **Pattern Fill**: Uses the shape of another trajectory without a gap to fill the selected gap. Use this tool only if there is a suitable marker with a trajectory similar to the one whose gap you wish to fill. This is typically the case when the trajectories originated from markers attached to the same segment, such as those attached to the ankle or heel. For more information, see Fill gaps with Pattern Fill on page 174.
- **Rigid Body Fill**: This option is the Nexus equivalent of the Replace 4 option, which is available in BodyBuilder. Use this option when a rigid or semi-rigid relationship exists between markers. For more information, see Fill gaps with Rigid Body Fill on page 175.
- **Kinematic Fill**: This option uses information about the connection of markers to segments in the labeling skeleton template (VST). For this option to be available, you may first need to run the **Kinematic Fit** pipeline operation, by clicking the **KinFit** button on the Nexus toolbar.For more information, see Fill gaps with Kinematic Fill on page 176.
- **Cyclic Fill**: For trials that contain captured data that is cyclic in nature (for example, when a motion on a treadmill or other repetitive motion is captured), this option uses patterns from a missing marker from earlier or later gait cycles to fill gaps. (If you want to modify the supplied operations, you can change and run the **Fill Gaps - Cyclic** pipeline operation instead.) For more information, see Fill gaps with Cyclic Fill on page 176.

> **Tip**
>
> For data with smaller gaps, you could use a processing pipeline containing the **Fill gaps (Woltring)** operation to automatically fill gaps.

8. In the chosen gap fill area, click the appropriate option:
   - To have Nexus fill the selected gap, click **Fill**.
   - To have Nexus fill all the gaps in the selected trajectory with the currently chosen type of gap filling, click **All**. This feature is particularly useful when, for example, you have 7 gaps in LASI and you want to fill them all with a **Rigid Fill** from the other pelvic markers (RASI, RPSI, LPSI).

- For **Pattern Fill** or **Rigid Body Fill**, you can click **Auto** before clicking **Fill** or **All**.

  When a gap is filled, Nexus reduces the entry in the **#Gaps** column by one and selects the next gap.
  When all gaps for the selected trajectory have been successfully filled, the entry for the trajectory is automatically removed from the **Trajectory** list and Nexus selects the next trajectory.

9. Repeat steps 4–8to fill all the gaps for all trajectories.
10. On the Nexus toolbar, click the **Save** button to save the trial. You can then perform any other processing that is required, such as running pipelines.

> **Tip**
>
> To save time, you can create custom pipelines that include the type of gap-filling operations that you commonly perform, such as Rigid fills for Pelvis and Head markers. For more information, see Example of creating a custom pipeline for gap-filling on page 179.

Use the gap filling options

The following information provides instructions on how to use the Vicon Nexus gap-filling options. (For detailed information about the Nexus gap-filling algorithms, see the FAQ What Gap Filling Algorithms are used in Nexus 2? on the Vicon website.)

Fill gaps with Spline Fill

1. Ensure the required gap is selected (see Manually fill gaps in trial data on page 170).
2. Under **Spline Fill**, in the **Maximum gap length** field, specify the maximum number of frames to spline fill (the default is 100 frames).
3. Click the appropriate button:
   - To have Nexus fill the selected gap, click **Fill**.
   - To have Nexus spline fill all the gaps in the selected trajectory, click **All**.

   A cubic spline interpolation operation fills the specified gaps.

Fill gaps with Pattern Fill

1. Ensure the required gap is selected (see Manually fill gaps in trial data on page 170).
2. Under **Pattern Fill**, click **Pick Source** or **Auto**:
   If you click Pick Source:
   a. The shape of the pointer changes to include a tooltip to guide you in the selection of markers in the **3D Perspective** view. Initially, a tooltip with the text **Pick source marker** hangs from the cursor.
   b. In the **3D Perspective** view, click a source marker whose trajectory is to be used to fill the gap of the target marker.
   A green dotted line is displayed to preview the shape of the trajectory if a pattern fill editing operation is run with the selected source model.
   Nexus displays the name of the source marker at the bottom of the **Pattern Fill** section, so you can confirm that you have selected the desired trajectory.
   c. In the **Pattern Fill** section, click **Fill** to have Nexus fill the selected gap (or **All** to fill all gaps in the selected trajectory), with the shape of the filled-in trajectory being similar to the trajectory of the source marker.

   If you click Auto:
   a. A suggested donor trajectory is listed in the space above the **Auto** button and the suggested fill is displayed in green in the **3D Perspective** window. You can also view the suggested fill options as colored dotted lines in the **Graph** view.
   b. Do one of the following:
      - To accept the suggested donor, click **Fill** or **All**; or
      - To reject the suggestion and manually pick a source, click the **Clear selected donor** button , click **Pick Source**, then click (or CTRL+click to multi select), the required trajectory, and then click **Fill** or **All**.

Fill gaps with Rigid Body Fill

This option is the Nexus equivalent of the **Replace 4** option, which is available in BodyBuilder.

Use it when a rigid or semi-rigid relationship exists between markers. For example:

- Pelvis (LASI, RASI, LPSI, RPSI)
- Head (LFHD, RFHD, LBHD, RBHD)

1. Ensure the required gap is selected (see Manually fill gaps in trial data on page 170).
2. In the **Rigid Body Fill** area, click **Pick Source** or **Auto**.
   If you click Pick Source:
   a. Click (or CTRL+click to multi select), the required trajectory
   b. Click **Fill** or **All**.

   If you click Auto:
   a. Three trajectories are listed in the space above the **Auto** button and the suggested fill is displayed in green in the **3D Perspective** window. You can also view the suggested fill options as colored dotted lines in the **Graph** view.
   b. Do one of the following:
      - To accept the suggested donor, click **Fill** or **All**; or
      - To reject the suggestion and manually pick a source, click the **Clear selected donor** button  , click **Pick Source**, then click (or CTRL+click to multi select), the required trajectory, and then click **Fill** or **All**.

Fill gaps with Kinematic Fill

This option uses information about the connection of markers to segments in the labeling skeleton (VSK).

1. To provide the required kinematic data, ensure you have clicked the **KinFit** button or run a pipeline operation that includes kinematic fitting.
2. Ensure the required gap is selected (see Manually fill gaps in trial data on page 170).
3. In the **Kinematic Fill** area, click **Pick Segment** and in the **3D Perspective** view select the segment from which the kinematics will be copied.
   The suggested fill is displayed in purple in the **3D Perspective** view. You can also view the suggested fill options as colored dotted lines in the Graph view.
4. Do one of the following:
   - To accept the suggested donor, click **Fill** or **All**; or
   - To reject the suggestion and try again, click the **Clear selected donor** button , click **Pick Segment**, then click (or CTRL+click to multi select), the required trajectory, and then click **Fill** or **All**.

Fill gaps with Cyclic Fill

1. Ensure you have captured a trial containing repetitive motion.
2. Select the gap/range that is to be filled (see Manually fill gaps in trial data on page 170).
3. In the **3D Perspective** view and the **Graph** of the marker component, preview the gap fill solution provided by Nexus.
4. If the suggested solution is acceptable, in the **Label/Edit Tools** pane, ensure **Gap Filling** is expanded.
5. In the **Cyclic Fill** area, select the required option:
   - **Fill** Fills the currently selected range
   - **All** Attempts to fill all gaps in the selected trajectory.

   The gap is filled with data based on other cycles of the same movement.

Automatically fill gaps in trial data

You can automatically fill gaps using the **Auto Gap Fill** button on the Nexus toolbar or the equivalent **Auto-Intelligent Gap Fill** pipeline in the **Pipeline** tools pane. This enables you to quickly fill gaps in your trial, without having to choose which fill method is best for each gap. You can do this in a one-click operation or, if you want to modify the supplied operations, by changing and running the **Auto Intelligent Gap Fill** pipeline.

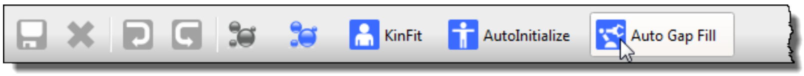
> **Tip**
>
> **Auto Gap Fill** attempts to fill all gaps in the trial data. For this operation to work well, you are likely to need to change the default property settings, including the gap size. For data containing large gaps, you are advised to review and manually fill the large gaps first, using the tools in the **Label/Edit Tools** pane (see Manually fill gaps in trial data on page 170), and/or cropping out any unwanted frames with large gaps that occur at the start or end of the trial. For information on cropping frames at the start and end of the trial data, see Crop trials on page 165.

To automatically fill gaps in trial data:

1. Display reconstructed and labeled markers in either of the following ways:
   - Ensure that a trial you have just captured has been reconstructed and labeled; or
   - From the **Data Management** tab, open a previously captured, reconstructed and labeled trial.
2. In the **Resources** pane, ensure Nexus is in **Offline** mode.
3. Display a **3D Perspective** view and optionally a **Graph** view.

   > **Tip**
   >
   > Using a split layout with both a **3D Perspective** and a **Graph** enables you to view the results of your editing actions and to select specific trajectories, frame ranges, and gaps in trajectories. Since the graphs indicate what changes have been made to the data and by which tool, you can immediately see the results of any edits you have made and determine if you want to change it again or adjust the data using one of the tools in the **Label/Edit Tools** pane.

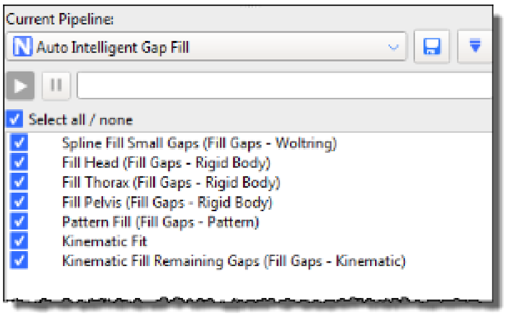
4. In the Nexus tool bar, click **Auto Gap Fill**.



Nexus assesses each gap one-by-one in series and fills all the gaps.

To use the Auto Intelligent Gap Fill pipeline:

1. On the **Tools** pane, click the **Pipelines** button.
2. From the **Current Pipeline** list, select the **Auto Intelligent Gap Fill** pipeline.



3. From the list of operations below, select the gap-filling operations that you want to use. If required, in the **Properties** pane, modify the relevant settings to suit your particular trial.
   For more information on an operation, either view its tooltip by clicking on its name and then hovering the mouse pointer over it or see the FAQ What gap-filling algorithms are used in Nexus 2? on the Vicon website.
4. Either click the **Run** button or use the **Auto Gap Fill** button on the tool bar, as described above, to run the pipeline with any changes you have made.
5. On the Nexus toolbar, click the Save button to save the trial. This will enable you to load the edited trial data. You can then perform any other processing, such as manually filling any remaining gaps.

You can run the pipeline either on an individual trial in the **Pipeline Tools** pane or on multiple files using the **Batch Processing Interface**.

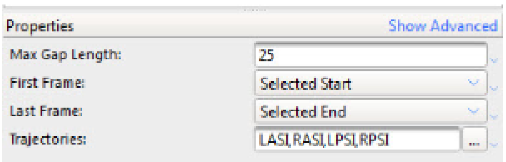
Example of creating a custom pipeline for gap-filling

The following steps describe how to make a custom pipeline for speeding up gap-filling operations. (For a reminder of how to create and run pipelines, see Work with pipelines on page 131.)

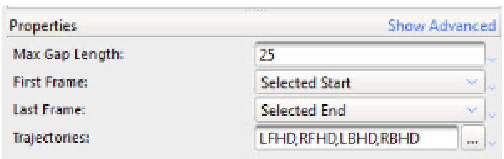
To create a custom gap-filling pipeline:

1. Create a new pipeline (for example, called **Head and Pelvis Fill**), which contains two **Rigid Body fill** operations.

2. Click on each operation in turn and in the **Properties** pane, customize the fills so that Pelvis and Head markers will be filled appropriately. For example:
   Pelvis Rigid Fill properties



   Head Rigid Fill properties



3. Save the pipeline.
4. Add a new button to the Nexus toolbar (**Window** menu > **Toolbar**) to give quick access to your custom pipeline.

   Now, with one mouse click, you can automatically fill any Pelvis and Head gaps.

Add events to trials

An event is a single action in the time span of a trial, for example, a foot contacting the floor during a walking trial, or a knee angle exceeding a particular angle.

Events are normally added to your trial data automatically as part of the capture workflow, when you run **Plug-in Gait Dynamic** or other dynamic pipelines that include **Detect Events From Forceplate** and **Autocorrelate Events**. For more information, see step 7 of Capture the required movement on page 137.

However, if events are not automatically detected, or are only partially detected, you will need to add them manually. This is likely to be necessary in the following scenarios:

- If you are not using a force plate; or
- The floor is not flat (ie, the movement being captured involves stepping, climbing, etc); or
- The trial includes movement that is not a cyclical gait movement (for example, some sports movements)

If your trial type does not allow the automatic addition of events, you can add them manually, using the timebar as described below.

For more information on working with events, see:

- Add events to trial data on page 180
- Delete events that are outside the region of interest on page 183
- Customize event identification mode on page 183
- Customize cycle visualization on page 183

Add events to trial data

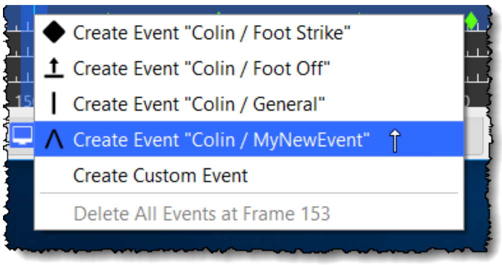
To add events to motion capture trial data:

1. From the **Data Management** tab, open the required trial, ensuring that it contains no gaps or other issues (see Review data quality on page 154 and Fill gaps in trial data on page 169).
2. Display the desired view (**3D Perspective**, **3D Orthogonal**, **Camera**, or **Graph**).
3. Enter event identification mode in either of the following ways:
   - On the time bar, click the **Enter Event Identification Mode** button; or.

- Press CTRL+E.

  The mouse pointer changes to a vertical arrow, to indicate that you are now in event identification mode.

4. On the time bar, zoom in or out on the timescale (right-click and drag) to adjust the time span of data displayed in the view pane and move the current time indicator (blue line) along the time bar to locate the desired frame.
5. Lock the event context by positioning the mouse pointer over the desired **Left**, **Right**, or **General** line on the selected frame (the line for a context is highlighted when the mouse pointer hovers over it) and pressing the UP ARROW or DOWN ARROW key. When the event context is locked, moving the mouse does not change the context.
6. Set a new event in the current context on the selected frame by either left-clicking or pressing ENTER and then selecting the desired command from the context menu
   - **Create Event Foot Strike** Creates a foot strike event at the selected frame.
   - **Create Event Foot Off** Creates a foot strike event at the selected frame.
   - **Create Event General** Creates a general event at the selected frame, represented by a vertical bar on the timeline. You cannot name general events and they are not saved to the event creation menu for future use.
   - **Create Custom Event** Creates a custom event at the selected frame that you can name, represented by a caret (^) on the timeline. When you create and name a custom event, it is added to the event creation menu, so you can select it again in future.



7. Navigate existing events in either of the following ways:
   - In the time bar controls, click the **Jump to the previous event** or **Jump to the next event** button; or
   - Press CTRL+LEFT ARROW or CTRL+RIGHT ARROW to go the previous or next event.

8. If required, you can:
   - Move an existing event to another frame by clicking in it with the left mouse button, holding the button down, dragging it left or right along the timeline.
   - Move the event at the current frame in the current context back or forward one frame by pressing ALT+Left- or Right-arrow key.
   - Delete existing events:
     - **Single event**: On the time bar, click the event or press ENTER, and from the displayed context menu select **Delete Event <Type>**.
     - **All events on current frame**: On the time bar, click the desired event or press ENTER, and from the displayed context menu select **Delete All Events at Frame x**.
     - **All events in trial**: On the time bar, right-click anywhere and from the displayed context menu select **Clear All Events**.

       Tip

       Note that you can also use the **Delete Timebar Events** pipeline operation (available on the **Pipeline Tools** pane, under **Events & Timebar**) to delete events from a selected range of frames.
9. When you have finished editing the current event context, unlock the event context by either moving the mouse over a different context or using the Up /Down arrow keys on the keyboard.
10. Repeat steps 5-9 for each event context you wish to edit.
11. When you have finished editing all events, exit event identification mode in any of the following ways:
    - In the timebar controls, click the **Enter Event identification mode** button again; or
    - Press CTRL+E; or
    - Press ESC

If you add gait cycle events before you crop a trial, you probably want to delete events that are outside the cropped region. (For information on cropping trial data, see Crop trials on page 165.)

Delete events that are outside the region of interest

If you crop a trial after you have added events to it, events may be retained in the discarded (cropped) region of the trial. To remove these events, do either of the following:

To delete events that are outside the region of interest:

1. Do either of the following:
   - Right-click on the time bar of the cropped trial and click **Delete Events Outside Region of Interest**; or
   - In the **Pipeline Tools** pane, from the **Events & Timebar** operations, select **Delete Timebar Events** and in the **Advanced Properties**, select whether to delete events inside or outside of the specified region.
2. Run the **Delete Timebar Events** operation to remove the unwanted events.

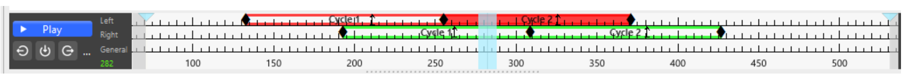
Customize event identification mode

To customize options that affect event identification mode:

1. Open the **Options** dialog box (press F7).
2. On the left of the dialog box, click **Time Bar**.
3. In the **Event Identification Mode** section on the right, select (or clear) **Focus on marker** and enter or select the markers to focus on for **Left** and **Right** contexts.

Customize cycle visualization

By default, all the cycles (gait or other) that have been defined by sets of time bar events, are displayed on the time bar.



This provides an easy way to identify:

- How many cycles are defined
- Where they are temporally
- Whether they are associated with force plate activity (indicated by solid color instead of colored outline only)
- How large they are (number of frames)

- Whether the left or right side is indicated:
  - Left = Red
  - Right = Green

To turn the visual display on or off or to change the default color options and other visual indicators, in the **Options** dialog box (press F7), on the left side click **Time Bar** and change the required properties on the right.

To turn on or off the display of the frame count, right-click on the time bar and select or clear **Show Cycle Frame Count**.

Modeling with Plug-in Gait

Vicon Nexus provides you with the following options for modeling:

- **Plug-in Gait** and the **Oxford Foot Model**. If you are new to modeling with Nexus, the Plug-in Gait model provides a good introduction: all the operations necessary to run the model are supplied with Nexus, easily accessible and ready to use. This option is described in this guide. For further details, see:

  - About the Plug-in Gait model on page 187
  - Plug-in Gait files installed with Vicon Nexus on page 189
  - How Plug-in Gait works on page 190
  - Take subject measurements for Plug-in Gait on page 191
  - Attach Plug-in Gait markers to a patient on page 199
  - Plug-in Gait Static pipeline on page 201
  - Plug-in Gait Dynamic pipeline on page 203

  In addition, a Vicon Nexus 2 tutorial video, *PlugIn Gait AI: Marker Locations and Subject Measurements*, is available on YouTube.

  > **Note**
  >
  > As the videos were recorded using an earlier version of Nexus 2, you may notice small differences in the user interface.

- **MATLAB**. If you are familiar with MATLAB, you can use the supplied examples to create your own custom models. For more information, see Modeling with MATLAB , in the *Vicon Nexus Reference Guide*.
- **Python**. If you are familiar with Python, you can use the supplied examples to create your own custom models. For more information, see Modeling with Python , in the *Vicon Nexus Reference Guide*.

Note

For simplicity, modeling operations are described as separate steps. However, if you want to run modeling operations automatically, you can add them (and any other operations required) to a post-capture pipeline that runs as soon as capture finishes. For more information see step 7 in Capture the required movement on page 137.

About the Plug-in Gait model

The Plug-in Gait model is Vicon's implementation of the Conventional Gait Model, which provides widely used and reliable full body kinematic and kinetic modeling, without the need for any customizations. It is based on the Newington-Helen Hayes gait model and has been validated through its frequent citation in peer-reviewed publications (see Plug-in Gait references on page 188).

Vicon Nexus includes the required pre-defined Plug-in Gait marker set and pipelines to enable you to produce outputs of the joint kinematics and kinetics for gait analysis patients. Plug-in Gait enables you to examine full, upper, or lower body joint kinematics and kinetics of patients. When you have run the relevant pipeline operations on your trials, you can produce gait analysis reports that conform to established clinical practices and import the resulting kinematic model of your subject (patient) into another software application, such as Vicon Polygon, for gait analysis.

It is assumed that you are familiar with Plug-in Gait, including the way the model works, where the markers are placed, and what the subject parameters mean. For example, Plug-in Gait directly calculates the kinematic model's joint centers from the measured XYZ marker positions on a frame-by-frame basis. Therefore, the lengths and orientations of the modeled segments are directly dependent on the marker positions. Accurate marker placement is therefore paramount. For guidance on marker placement, see Attach Plug-in Gait markers to a patient on page 199.

The model uses three or more points to define each segment. For all segments distal to the pelvis, one of the points used is the joint center that defines the proximal joint for the segment, for example, one of the three defining points for the tibia is the knee joint center.

Operations relating to modeling with Plug-in Gait are available in the **Data Processing** operations in the **Pipeline Tools** pane.

For in-depth descriptions of Plug-in Gait models and templates, and details of the calculations performed by Plug-in Gait, see the *Plug-in Gait Reference Guide*.

Before using Plug-in Gait, be sure to read the Plug-in Gait disclaimer on page 188.

Plug-in Gait references

The following research publications provide supporting information on the scientific basis and validation of Plug-in Gait:

- DAVIS R., OUNPUU S., TYBURSKI D., and GAGE J. R. *A gait analysis collection and reduction technique* . Human Movement Science, 10, p575–587, (1991).
- KADABA M. P., RAMAKRISHNAN H. K., and WOOTTEN M. E. *Measurement of lower extremity kinematics during level walking*. Journal of Orthopaedic Research, 8, p383–392, (1990).
- WOLTRING, H. J. *A FORTRAN package for generalized, cross-validatory spline smoothing and differentiation*, Advances in Engineering Software, 8, p104–113, (1986).
- VAN DEN BOGERT, T. *Practical Guide to Data Smoothing and Filtering.* See http://isbweb.org/software/sigproc/bogert/filter.pdf

Plug-in Gait disclaimer

You are entirely responsible for the use you make of the software. In particular, the software is not intended to replace the clinical skill of a medical practitioner or his/her independent professional judgment of individual clinical circumstances to make a diagnosis and/or determine a patient's treatment.

The software should only be used by those who have been appropriately trained in its operation, functions, capabilities, and limitations, and in any event should not be relied upon, by itself, as the sole method of making any diagnosis or determining any treatment.

Plug-in Gait files installed with Vicon Nexus

The following table lists the Plug-in Gait files that are automatically installed with Nexus. For more detailed information about the labeling skeleton template (*.vst*) files, see the *Plug-in Gait Reference Guide*.

File name	Description
PlugInGait FullBody Ai.vst	Full body labeling skeleton template
PlugInGait LowerBody Ai.vst	Lower body labeling skeleton template
Marker set	
PlugInGait FullBody Ai.mkr	Markers defined in full body model
PlugInGait LowerBody Ai.mkr	Markers defined in lower body model

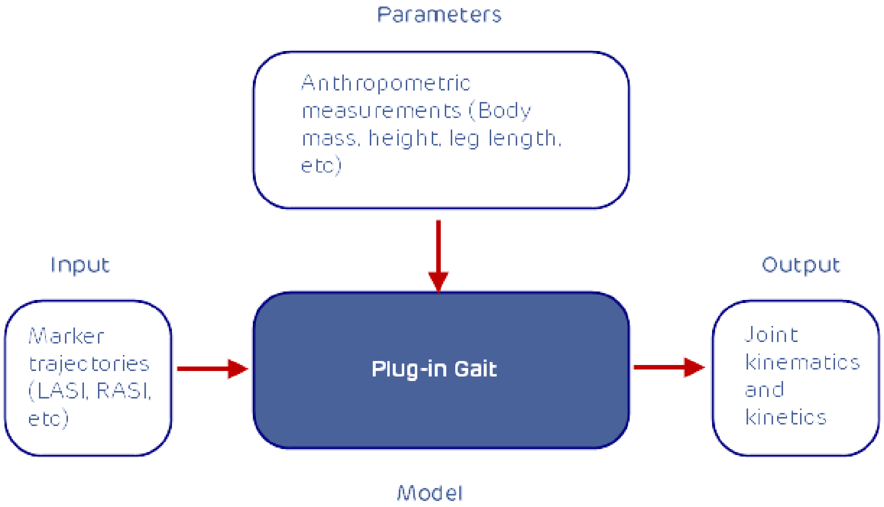
By default, the supplied files are installed in the following folder:

C:\Program Files (x86)\Vicon\Nexus2.#\ModelTemplates

How Plug-in Gait works

To work with Plug-in Gait, you enter subject-specific measurements into a pre-defined template in Vicon Nexus, place a Plug-in Gait marker set on a patient, capture the required trials, and then use Plug-in Gait pipelines in Nexus to produce outputs of the joint kinematics and kinetics for the patient.

From the real marker trajectories, Plug-in Gait generates virtual marker trajectories that represent kinematic and kinetic quantities (angles, moments, etc) and representations of the modeled segments. The model also outputs data that is used to define the positions of meshes (representing bones) which can subsequently be displayed in Vicon Polygon (for further details, see the Vicon Polygon documentation).



For detailed information about modeling with Plug-in Gait, see *Plug-in Gait kinematic and kinetic calculations* in the *Plug-in Gait Reference Guide.*

Before you begin capturing, make sure you understand the types of outputs that Plug-in Gait produces, so that you can determine the appropriate model to use for your requirements. If you are unsure which is the most appropriate model to use, see *Models and templates* in the *Plug-in Gait Reference Guide.*

Take subject measurements for Plug-in Gait

Plug-in Gait requires a set of subject measurements to enable it to directly calculate kinematics (angles) and kinetics (forces, moments, and powers) from the measured XYZ marker positions. Some of the measurements you must add to the subject node are required for the model to run, and some are optional:

- **Required subject measurements** You must add the required measurements to the subject node for your patient. Required measurements are listed in the following topics and are highlighted in pink in the **Properties** pane for the selected subject in Vicon Nexus.
- **Optional measurements** You can add any optional measurements or leave them for Plug-in Gait to calculate.

The following topics list the measurements needed for each type of model:

- Subject measurements for Plug-in Gait lower body model on page 192
- Subject measurements for Plug-in Gait full body model on page 195

Tip

You may find it helpful to take these measurements before you begin the subject preparation process in Vicon Nexus. You can use the **Measure** column in the tables in these sections to record the measurements you take for your patient, so you will have them to hand when you are ready to add them to your chosen labeling skeleton template (.vst file). You may want to photocopy the blank form for each patient before recording your measurements.To avoid the markers getting in the way, take the measurements before attaching markers to your patient.

Subject measurements for Plug-in Gait lower body model

The following table identifies the subject measurements that are used as inputs for running a Plug-in Gait lower body model.

Optional measurements (and/or those calculated by the model) are marked with an asterisk in the **Name** column.

Name	Description	Measure Left	Measure Right
Body Mass	Patient mass.	_____ kg	
Height	Patient height.	_____ mm	
*Inter ASIS distance	The model calculates this distance based on the position of the LASI and RASI markers. If you are collecting data on an obese patient and cannot properly place the ASIS markers, place those markers laterally and preserve the vector direction and level of the ASIS. Palpate the LASI and RASI points and manually measure this distance, then input into the appropriate field.	_____ mm	
Leg Length	Full leg length, measured between the ASIS marker and the medial malleolus, via the knee joint. Measure with patient standing, if possible. If the patient is in the crouch position, this measurement is NOT the shortest distance between the ASIS and medial malleoli, but rather the measure of the skeletal leg length, so if a patient cannot straighten his/her legs, take the measurement in two pieces: ASIS to knee and knee to medial malleolus.	_____ mm	_____ mm

Name	Description	Measure Left	Measure Right
*ASIS-Trochanter Distance	ASIS-greater trochanter distance is the vertical distance, in the sagittal plane, between the ASIS and greater trochanter when the patient is lying supine. Measure this distance with the femur rotated such that the greater trochanter is positioned as lateral as possible.	____ mm	____ mm
Knee Width	The medio-lateral width of the knee across the line of the knee axis. Measure with patient standing, if possible.	____ mm	____ mm
Ankle Width	The medio-lateral distance across the malleoli. Measure with patient standing, if possible.	____ mm	____ mm
*Tibial Torsion [1]	The angle between the knee flexion and the ankle dorsi-plantar axes. The ankle is usually externally rotated with respect to the knee flexion axis. If you are using a KAD, and the medial malleoli markers are attached to the patient, Plug-in Gait calculates the tibial torsion automatically.	____ deg	____ deg
*Sole Thickness Delta[2]	The difference in the thickness of the sole at the toe and the heel. A positive sole delta indicates that the patient's heel is raised compared with the toe.	____ mm	____ mm
*Thigh Rotation	When a KAD is used, this value is calculated to account for the position of the thigh marker. By using the KAD, placement of the thigh marker in the plane of the hip joint center and the knee joint center is not crucial. If you do not use a KAD, this value is zero as the model assumes that the thigh	____ deg	____ deg

Name	Description	Measure Left	Measure Right
	marker is placed exactly in the plane of the hip joint center and the knee joint center. This value is calculated for you.		
*Shank Rotation	Similar to the Thigh Rotation. This value is calculated if a KAD is present and removes the importance of placing the shank marker in the exact plane of the knee joint center and ankle joint center. If you do not use a KAD, this value is zero. It is calculated for you.	_____ deg	_____ deg

1. The tibial torsion measurement is only needed if you are using a KAD.
2. The **Sole Thickness Delta** subject measurement is used if the subject is wearing high-heeled footwear but can keep their foot flat within the shoe. For more information, see Sole Thickness Delta subject measurement on page 194.

After you have recorded the measurements for your patient, you can create a subject node for the patient and enter the measurements into Nexus as described in Create a new subject from a template on page 110.

Sole Thickness Delta subject measurement

The **Sole Thickness Delta** subject measurement is used when the patient is wearing shoes or orthoses so that although the patient achieves foot flat with respect to the footwear, the sole of the footwear introduces a vertical difference between the toe and the heel. When this is the case, you can either:

- Compensate for the sole delta by carefully placing the heel marker with respect to the toe marker and in the **Static Settings** dialog box, ensure the check boxes for **Left Foot** and/or **Right Foot** are cleared.

or

- Measure the difference in the thickness of the sole at the toe and the heel, in the **Static Settings** dialog box, select the check boxes for **Left Foot** and/or **Right Foot** and then in the **Properties** section at the bottom of the **Subjects Resources** pane, enter the delta value in the **SoleDelta (mm)** box. The height of the heel marker used in the calculations will then be adjusted accordingly.

Subject measurements for Plug-in Gait full body model

The following table lists the subject measurements that are used as inputs for running Plug-in Gait for full body modeling.

Optional measurements are marked with an asterisk in the **Name** column.

Name	Description	Measure Left	Measure Right
Body Mass	Patient mass.	_____ kg	
Height	Patient height.	_____ mm	
*Inter-ASIS distance	ASIS-ASIS distance is the distance between the left ASIS and right ASIS. This measurement is only needed when markers cannot be placed directly on the ASIS, for example, in obese patients.	_____ mm	
*Head Offset	Patient head offset in degrees.Only required if head is not level (calculated after running the Plug-in Gait model).	_____ deg	
Leg Length	Full leg length, measured between the ASIS marker and the medial malleolus, via the knee joint. Measure with patient standing, if possible. If the patient is standing in the crouch position, this measurement is NOT the shortest distance between the ASIS and medial malleoli, but rather the measure of the skeletal leg length.	_____ mm	_____ mm
*ASIS-Trochanter Distance	ASIS-greater trochanter distance is the vertical distance, in the sagittal plane, between the ASIS and greater trochanter when the patient is lying supine. Measure this distance with the femur rotated such that the greater trochanter is positioned as	_____ mm	_____ mm

Name	Description	Measure Left	Measure Right
	laterally as possible. If this value is not entered, a regression formula is used to calculate the hip joint center. If this value is entered, it will be factored into an equation which represents the hip joint center.		
Knee Width	The medio-lateral width of the knee across the flexion axis. Measure with patient standing, if possible.	_____ mm	_____ mm
Ankle Width	The medio-lateral distance across the malleoli. Measure with patient standing, if possible.	_____ mm	_____ mm
*Tibial Torsion 1	The angle between the knee flexion and the ankle dorsi-plantar axes. The ankle is usually externally rotated with respect to the knee flexion axis. The sign convention is that if a negative value of tibial torsion is entered, the ankle flexion/extension axis will be adjusted from the KAD's defined position to a position dictated by the tibial torsion value. If you are using a KAD, and the medial malleoli markers are attached to the patient, Plug-in Gait calculates the tibial torsion automatically.	_____ deg	_____ deg
*Sole Thickness Delta[2]	The difference in the thickness of the sole at the toe and the heel. A positive sole delta indicates that the patient's heel is raised compared with the toe.	_____ mm	_____ mm
*Thigh Rotation	When a KAD is used, this value is calculated to account for the position of the thigh marker. By using the KAD, placement of the thigh marker in the plane of the hip joint	_____ deg	_____ deg

Name	Description	Measure Left	Measure Right
	center and the knee joint center is not crucial. If you do not use a KAD, this value is zero as the model assumes that the thigh marker is placed exactly in the plane of the hip joint center and the knee joint center. This value is calculated for you.		
*Shank Rotation	Similar to the Thigh Rotation. This value is calculated if a KAD is present and removes the importance of placing the shank marker in the exact plane of the knee joint center and ankle joint center. If you do not use a KAD, this value is zero. It is calculated for you.	____ deg	____ deg
Shoulder Offset	Vertical distance from the center of the glenohumeral joint to the base of the marker on the acromion clavicular joint. Some researchers have used the (anterior /posterior girth)/2 to establish a guideline for the parameter.	____ mm	____ mm
Elbow Width	Width of elbow along flexion axis (roughly between the medial and lateral epicondyles of the humerus).	____ mm	____ mm
Wrist Width	Anterior (palm side)/Posterior (back) thickness of wrist at position where wrist marker bar is attached.	____ mm	____ mm
Hand Thickness	Anterior/Posterior thickness between the dorsum and palmar surfaces of the hand, at the point where you attach the hand marker.	____ mm	____ mm

1. The tibial torsion measurement is only needed if you are using a KAD.

2. The **Sole Thickness Delta** subject measurement is used if the subject is wearing high-heeled footwear but can keep their foot flat within the shoe. For more information, see Sole Thickness Delta subject measurement on page 194.

After you have recorded the measurements for your patient, you can create a subject node for the patient and enter the measurements into Vicon Nexus as described in Create a new subject from a template on page 110.

Attach Plug-in Gait markers to a patient

After adding the patient's measurements to the subject node (see Take subject measurements for Plug-in Gait on page 191), you attach the markers to the patient.

Vicon cameras capture the movement of the retroreflective markers rather than the body to which they are attached. To enable Nexus to determine the movement of the subject's underlying skeleton, ensure that the subject's clothing will not occlude markers or by cause the markers to move excessively. If possible, attach the markers directly to the patient's skin. During capture, ensure that there is nothing else reflective in the capture volume.

To attach markers to a patient:

1. Select the appropriate size and quantity of Vicon markers to be used. Typically, 14 mm markers are used for gait analysis studies.
2. Securely attach the Vicon markers to the patient in accordance with the marker arrangement defined in your chosen template. A front view of the most commonly used set (for Plug-in Gait lower body modeling) is shown in the following image. The left lower body markers are not labeled; place markers on the left side in a similar way to those on the right.

   Important

   Note that, as shown in the following image, some asymmetry is desirable as it helps the auto labeling routine distinguish right from left. In a lower body marker set, you can place the THI and/or TIB markers asymmetrically. For a full body set, you can place the THI, TIB, UPA and FRM markers asymmetrically. Similarly, avoid symmetrical placement of marker clusters or groups of markers and also ensure markers are asymmetrical within each cluster/group.

Tip

- The THI and TIB markers anterior-posterior position is critical for identifying the orientation of the knee and ankle flexion axis.
- If you are using a single sacral marker (SACR) in a Plug-in Gait lower body or full body marker set, attach the SACR marker instead of the two PSIS markers.
- Because you need at least three markers to obtain six degrees of freedom, best practice is to place four markers on the pelvis (LASI /RASI/LPSI/RPSI), so that even if one of the four markers is occluded during motion capture, the required three markers will still be visible to the cameras.

For detailed descriptions and images of marker positions, see *Models and templates* in the *Plug-in Gait Reference Guide*.

You can now capture a calibration trial and reconstruct the markers (see Calibrate a labeling skeleton on page 113).

Plug-in Gait Static pipeline

As described in step 5 of Review and finalize a calibration of a labeling skeleton using a ROM trial on page 120, to enable Plug-in Gait to calculate the static outputs based on the measured marker positions, you can run a **Plug-in Gait Static** pipeline on the reconstructed and labeled static frames of either a ROM trial or a static trial.

> **Important**
>
> The Plug-in Gait Static pipeline is intended to calculate subject-specific offsets for use as inputs to the Plug-in Gait Dynamic pipeline. If joint angles are required, you must run the Plug-in Gait Dynamic pipeline (page 203) in addition to the static pipeline, even for a stationary subject.

Process Static Plug-in Gait Model properties

The following table describes the properties you can specify for the **Process Static Plug-in Gait Model** pipeline operation:

Property	Description
First Frame	Specify the first frame to be used for processing.
Last Frame	Specify the last frame to be used for processing.
Marker Diameter	Specify the diameter of the markers used, assuming they are applied to the skin's surface. The default is 14mm.
Anterior Thorax Tilt is Positive	Select this option to specify that the anterior thorax tilt is positive. Clear this option to specify that the anterior thorax tilt is negative.
Output Joint Centers	Select this option to visualize the joint centers that are calculated by Plug-in Gait.

Property	Description
Assume Horizontal In this section, you can configure Plug-in Gait to assume that certain markers are at the same height from the floor during the static trial. Set the appropriate option to enable Plug-in Gait to compensate for your patient's stance during the static trial. **Important:** Set the appropriate foot option if you specify the Sole Thickness Delta subject measurement because the patient is wearing shoes or orthoses that introduce a vertical difference between the toe and the heel (for details, see Subject measurements for Plug-in Gait lower body model on page 192).	
Left Foot	Plug-in Gait assumes that the LTOE and LHEE markers are level with one another. Select this option if the patient's left foot was flat with the toe and heel in contact with the floor, even if the left toe (LTOE) and heel (LHEE) markers were not level with one another.
Right Foot	Select this option if the patient's right foot was flat with the toe and heel in contact with the floor, even if the right toe (RTOE) and heel (RHEE) markers were not level with one another.
Head	Select this option if the patient's head was held level during the static trial, even if the rear of head markers (LBHD,RBHD) could not be placed level with the front of head markers (LFHD, RFHD).

Plug-in Gait Dynamic pipeline

To enable Plug-in Gait to finalize trial data, including filtering the data, detecting gait events, and producing a *.c3d* file for export from Nexus, you can run a **Plug-in Gait Dynamic** pipeline on the reconstructed and labeled movement trials.

Before you run the pipeline on your movement trials, ensure you have gap-filled and if necessary cropped the trials (see Review trials and fill gaps on page 152).

If required, you can change the properties for each of the operations that are included in the **Plug-in Gait Dynamic** pipeline. You can also run them individually, and/or run other operations at this stage too. To help you understand the different operations typically run on Plug-in Gait movement trials and the options available, each operation is described in the following topics, together with the **Delete Unlabeled Trajectories** pipeline operation, which you may also find useful:

- Run the Dynamic Plug-in Gait pipeline on page 204
- Delete Unlabeled Trajectories pipeline operation on page 205
- Filter Trajectories - Woltring pipeline operation on page 205
- Detect Events From Forceplate pipeline operation on page 206
- Autocorrelate Events pipeline operation on page 207
- Process Dynamic Plug-in Gait Model pipeline operation on page 208
- Cross-plate foot strikes on page 210
- Export C3D on page 213

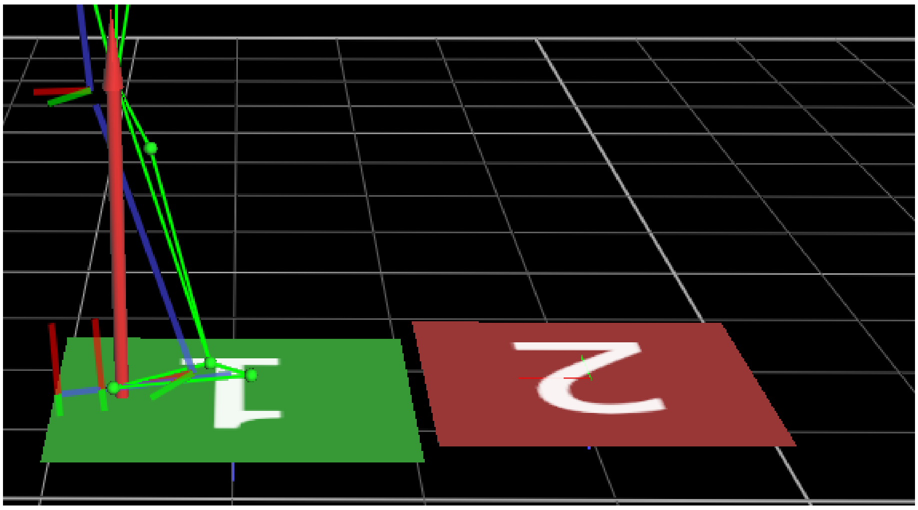
Tip

When you are familiar with these operations, you can run all of the data processing in a single step by creating your own custom pipeline and adding all the operations described in this section to it, as explained in Create a pipeline on page 134. When you create your new pipeline, ensure that you include the operations in the above order.

Run the Dynamic Plug-in Gait pipeline

To run the Dynamic Plug-in Gait pipeline:

1. On the **Data Management** tab, ensure the dynamic trial you have gap-filled is open (for details, see Fill gaps in trial data on page 169).
2. On the **Pipeline** tab of the **Tools** pane, from the **Current Pipeline** list, select **Dynamic Plug-in Gait**.
3. Ensure that the required operations are selected and that you have configured their properties appropriately (see below for details).
4. Click the Play button.
   A green check mark appears in the list to the left of each completed operation.
   In the **3D Perspective** view, the axis for each segment is displayed, together with the **Foot Strike** and **Foot Off**.

5. In the **SubjectsResources** pane, expand the subject's tree and ensure that the model outputs are listed.
6. To save the data in *.c3d* format, on the Vicon Nexus toolbar, click the Save button.

If you don't need to perform any further processing, you can now export your data. For more information, see Export trial data on page 233.

Delete Unlabeled Trajectories pipeline operation

This pipeline operation runs on a fully labeled trial to delete any remaining unlabeled trajectories. To choose the maximum length of unlabeled trajectory to delete, select the **Delete using Max Length** option and specify the **Max Length to Delete**.

Filter Trajectories - Woltring pipeline operation

This pipeline operation filters the data using the Woltring filter to ensure smooth trajectories for calculating kinetics. This routine is based on a fifth-order spline-interpolating function (for details on this function and its filtering algorithms, see Plug-in Gait references on page 188).

In the **Properties** pane for this operation, you can specify the following settings:

Property	Description
First Frame	First frame to which filtering is applied
Last Frame	Last frame to which filtering is applied
Filter mode	Choose between: - **GCV** The filtering routine makes an automatic estimate of the noise to find the optimal smoothing parameter. - **MSE** You can change the level of smoothing by entering a different value for the **Smoothing** option below.
Trajectories	Enables you to select the trajectories to filter. Can be All trajectories, All labeled trajectories, or Linked trajectories.
Smoothing	Enables you to specify the level of smoothing for the **MSE** option.

Detect Events From Forceplate pipeline operation

This pipeline operation automatically detects gait cycle events such as footstrikes and adds them to the time bar throughout the trial using vertical ground reaction forces (GRFs) measured by a force plate connected to the Vicon system. The operation includes the option to set the force threshold and the label of the markers attached to the front and back of the foot for both sides.

In the **Properties** pane for this operation, you can specify the following settings:

Property	Description
First Frame	First frame in which events are detected
Last Frame	Last frame in which events are detected
Force Threshold (N)	The force threshold for automatically detecting heel strike and toe off events. The default is 20.
Left anterior marker label	Marker used for event detection. You are recommended to leave this at the default setting.
Left posterior marker label	Marker used for event detection. You are recommended to leave this at the default setting.
Right anterior marker label	Marker used for event detection. You are recommended to leave this at the default setting
Right posterior marker label	Marker used for event detection. You are recommended to leave this at the default setting.

Autocorrelate Events pipeline operation

This pipeline operation detects the pattern of the tracked marker at the set events and defines these events for the rest of the trial. The available parameters are: marker being tracked; the X, Y, or Z component of the marker; and the position, velocity, or acceleration of the marker. Run this operation after **Detect Events from Forceplate** or manually entering events on the timebar.

In the **Properties** section for this operation, you can specify the following settings:

Property	Description
First Frame	First frame in which events are detected
Last Frame	Last frame in which events are detected
Only after existing	If a force plate is used and you select this option, Vicon Nexus automatically correlates force plate data events from the time of the first strike on the force plate and forward. If you do not select this option, Vicon Nexus automatically correlates events both forward and backward from the first force plate strike. If a force plate is not used, you can still select this option, but you must make a thorough visual check of the events placed.
Marker (Left and Right)	Enables you to change the marker used for correlation if necessary.
Component (Left and Right)	For walking trials, leave the Left and Right values at the default value (Z). For running and other motor tasks, change as appropriate.
Derivative (Left and Right)	For walking trials, leave the Left and Right values at the default value (Position). For running and other motor tasks, change as appropriate.

Process Dynamic Plug-in Gait Model pipeline operation

This pipeline operation runs the dynamic Plug-In Gait model on the active subject of the current trial.

In the **Properties** section for this operation, you can specify the following settings:

Property	Description
First Frame	First frame on which the operation is run
Last Frame	Last frame on which the operation is run
Marker diameter (mm)	Ensure the specified value corresponds to the size of the markers attached to your patient.
Reaction reference frame	Determines which reference frame is used for reporting joint moments: - **Proximal segment**: Reports all moments in the reference frame of the proximal segment. - **Distal segment**: Reports all moments in the reference frame of the distal segment (for example, the tibia segment for the knee joint). - **Global frame**: Reports the moment vector relative to the laboratory coordinate system.
Power Output	How Plug-in Gait is to output the power data: - **Standard**: The joint power expressed as a single number (scalar). - **Individual Contributions from XYZ**: The joint power expressed as individual X, Y, and Z planes in the segment's coordinate system
Anterior Thorax Tilt is Positive	Select this option to specify that the anterior thorax tilt is positive. Clear this option to specify that the anterior thorax tilt is negative.

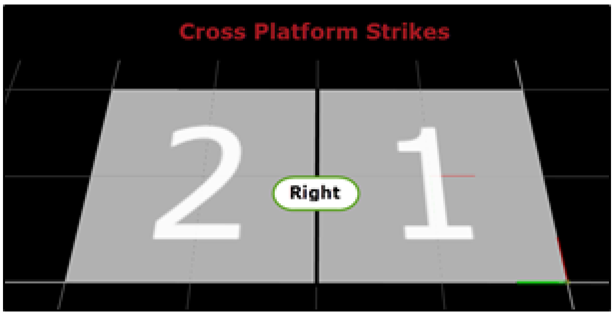
Property	Description
Output Joint Centers	Select this option to visualize the joint centers that are calculated by Plug-in Gait.
Allow cross-plate strikes	Select this option when a foot may be in contact with more than one force plate simultaneously; for example, the heel may land on the far side of one plate and then the toes roll onto the near edge of the subsequent plate. If you select this option, reactions from the two plates are combined during processing, with the plate that is struck first being considered the primary. The forces reported by each plate are added together; the moments are also combined by referring the moment from the secondary plate back to the origin of the primary plate. In effect, the two plates are treated as a single force plate with the same origin as the primary plate. For more information, see Cross-plate foot strikes on page 210. Although every effort has been made to ensure the mathematical combination of forces and moments between two plates is correct, it is the responsibility of the operator / analyst to review the outcomes produced by this process and ensure they are correct. Use of this option in clinical assessments is strictly at the discretion of the operator/analyst.
Radius of Gyration: Pelvis and Thorax	The default radius of gyration for the specified segments. The number is scaled to the length of the segment. The Plug-in Gait default setting (0.31) means that the radius of gyration is 31 percent of the segment length. Only change this setting if you are sure that the default value is incorrect.
Version	Plug-in Gait version number.

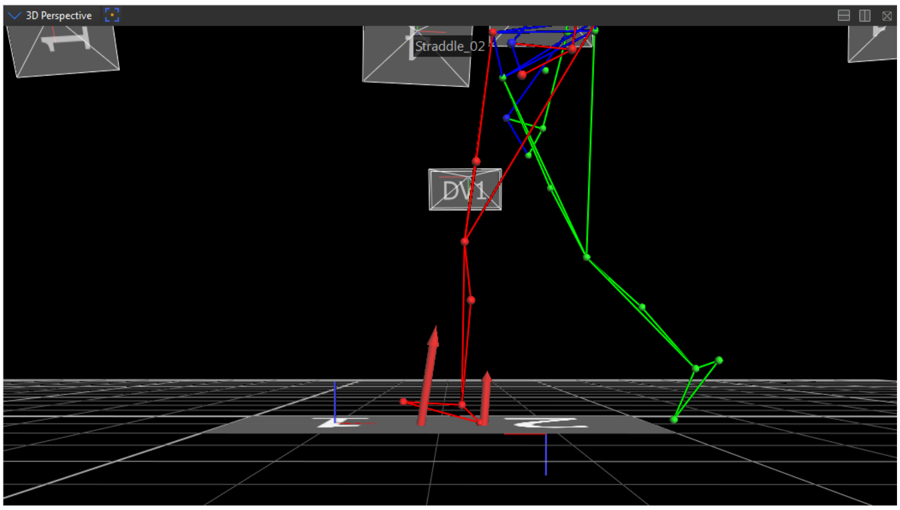
Cross-plate foot strikes

To perform an inverse reaction calculation from a foot strike, forces and moments recorded by a force plate must:

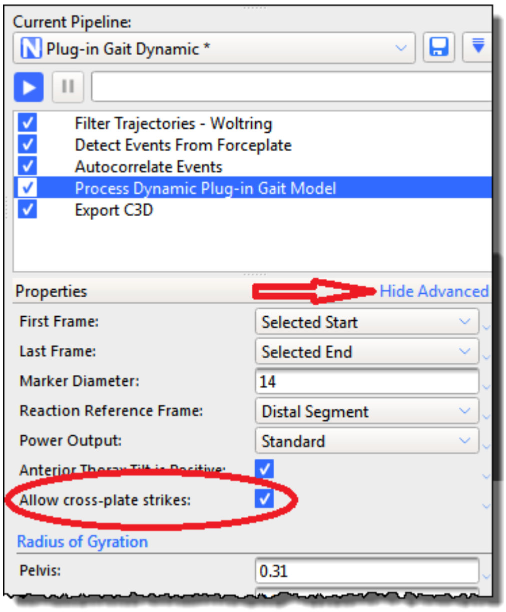
- Be assigned to a single context (left or right)
- Represent the full forces and moments produced by the subject

In some instances, a foot may be in contact with more than one force plate simultaneously; for example, the heel may land on the far side of one plate and then the toes roll onto the near edge of the subsequent plate.





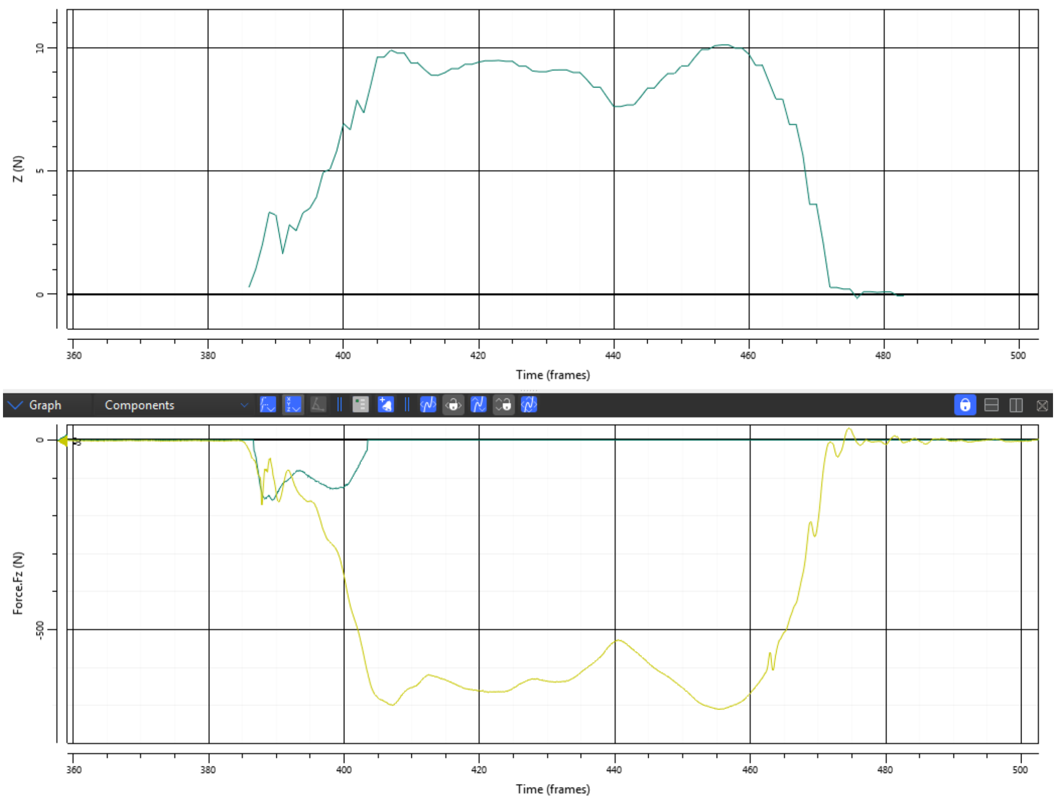
Dynamic Plug-in Gait includes an Advanced option to permit calculations based on these cross-plate strikes.



Reactions from the two plates are combined during processing, with the plate that is struck first being considered the primary. The forces reported by each plate are added together; the moments are also combined by referring the moment from the secondary plate back to the origin of the primary plate. In effect, the two plates are treated as a single force plate with the same origin as the primary plate.

Dynamic Plug-in Gait can automatically detect which foot is activating a force plate by checking whether the origin of the segment is above the plate. To account for cross-plate strikes, the activating area is expanded beyond the bounds of the plate by the length of the segment (that is, a foot is considered to be above the plate if the ankle joint center is within a foot-length of the plate boundary).

You may also set the foot strike manually (see Add events to trials on page 180 and also Automatically assess foot strikes on page 146).

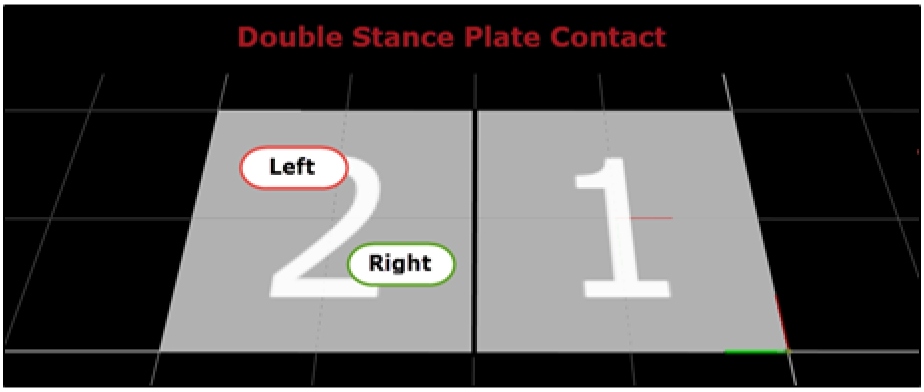


Note that the existing restriction that there must be only a single foot in contact with any given force plate still applies.

Invalid strike types that do not benefit from the Allow cross-plate strikes option

Only cross plate strikes can benefit from the **Allow cross-plate strikes** option; do not use other invalid strike types for kinetic calculation. The other invalid strike types to which Allow cross-plate strikes does *not* apply include:

- **Double stance plate contacts**
  Multiple feet simultaneously in contact with a single plate.
  In the following example, the left and right foot of a subject contact Force Plate 1 at the same time.

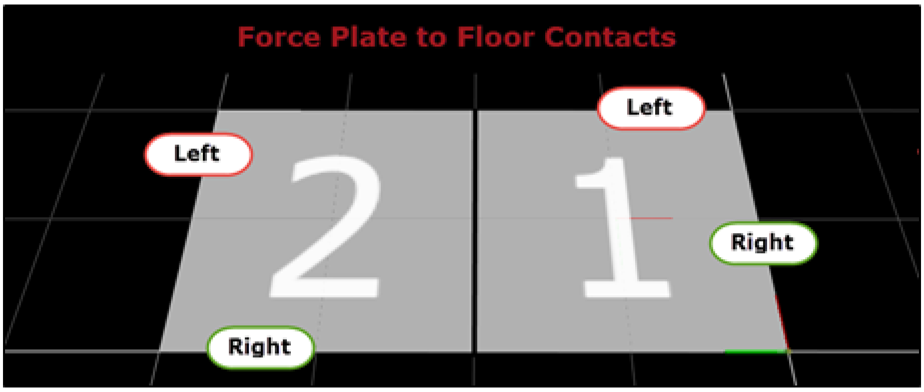


Problem: An inability to assign forces to context

- Force plate to floor contacts

  A single foot strike that transitions from an initial contact with the force plate to the lab floor or begins on the floor and transitions to a force plate.

  In the following example, the heel strikes the force plate but the foot rolls forward off the force plate and the later stages of foot contact prior to toe off are in contact with the floor.



Problem: Full forces / moment are not recorded as the some of these are applied to the floor and not the plate.

Export C3D

This pipeline operation exports the current state of the processed data to a .c3d file. You can then import the data into other software for further processing or report generation. For example, in Vicon Polygon you can visualize the trajectories, kinematic model elements, and kinetics data. If you manually import the corresponding .vst file into Polygon, it also visualizes the bones. For more information, see Configure the Export C3D operation on page 238.

Work with digital video files

If you are using reference video cameras (for example, Vicon Bonita Video or Vicon Vue cameras), data from these cameras is captured simultaneously with optical motion data from the Vicon optical cameras.

As video files are large and can take a lot of time to process, to save time and space on the hard drive, you normally set up the video cameras to save captured video data to separate, dedicated drives on the host PC, as .vvid files. (For information on video camera setup, see Configure video cameras for digital video capture on page 61 and the PDF PC setup for Vicon systems). Note that you cannot view the video in Nexus until the files are transcoded.

To view the video files, after you have captured and processed your trials, at a suitable pause in the workflow, you transfer the video files to the host PC, simultaneously transcoding the files to a format that is viewable in Nexus.

For more information, see:

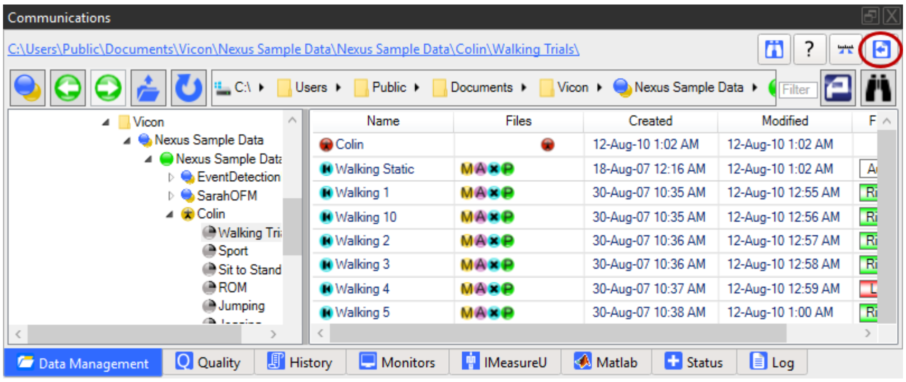
- About transferring and transcoding on page 215
- Transfer and transcode digital video files on page 217
- De-interlace AVI files on page 220

About transferring and transcoding

Tip

Because of the size of video files, transferring and transcoding takes up time and computing resources, so is best done as a batch process on a number of video files during a break or after the day's capture session ends. Ensure you transfer, transcode and then delete the .vvid files regularly to avoid filling up your SSDs.

To enable you to transfer and transcode reference video files, Nexus provides the **Show File Transfer/Batch Processing interface**, which you access via a button on the **Data Management** toolbar at the top of the **Data Management** tab.



You can transcode and transfer reference video files to the Nexus host PC that have been produced by one or more Vicon Vue cameras, Bonita Video cameras, or supported Basler cameras and recorded to separate drives on the host PC:

- Vicon video cameras are easiest to configure and offer higher frame rates than other supported video cameras.
- Supported Basler cameras capture video data in raw format and stream this data directly to a hard drive. This allows Nexus to capture video as quickly as possible while minimizing the chances of dropped video frames during collection; however raw Basler video files are very large, making storage on local hard drives difficult.

Comparison of vvid files and transfer rates for supported video cameras

All values are approximate and are provided for guidance only.

Camera type	Width	Height	Frame rate	MB\Sec
Vue (no windowing) Vue (with windowing)	1920 1280	1080 720	60 120	105 105
Bonita 720c	1280	720	127	111
Bonita 480m	640	480	360	105
Basler piA640 210 gc	648	488	210	63
Basler piA1000 48gc\m	1004	1004	48	46
Basler A602gc	656	490	100	30
NTSC DV	720	486	29.97	32
PAL DV	720	576	25	31

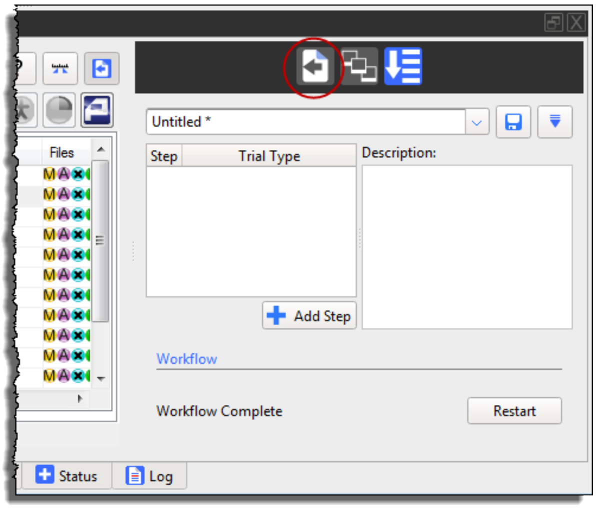
Video files are large, so as part of the transfer process, you normally select a codec (Compressor/Decompressor) format to reduce the video file size. The selection of an appropriate codec and codec settings ensures the maximum reduction in file size, while minimizing any reduction in video quality.

Transfer and transcode digital video files

If a video file within the currently open trial has already been transferred and/or transcoded, Nexus automatically loads the new version of the file.

To transfer and transcode reference video files:

1. On the **Data Management** tab, in the **Show File Transfer/Batch Processing interface** (see About transferring and transcoding on page 215), click the **File Transfer** button.



   The list in the **Trial** column includes all digital video capture trials created during video capture.

2. From the list, specify the video files to be transferred in any of the following ways:
   - Click the corresponding check box to select a specific trial.
   - Click the **Select All** button to select all trials in the list.
   - Click **Select None** button to clear all previously selected trials.
3. In the **File Transfer** area, from the **Video Compression** drop-down menu either leave **None** (the default setting) or select one of the supported codecs for compressing the selected video files prior to transfer. Vicon recommends that you use the FFDShow codec, which you can download from the Third-Party Plugins page on the Vicon website. For information on installing and setting the correct options for the FFDShow codec, see Install the FFDShow codec on page 219.

> **Tip**
>
> Nexus remembers the last codec selected from the drop-down list.

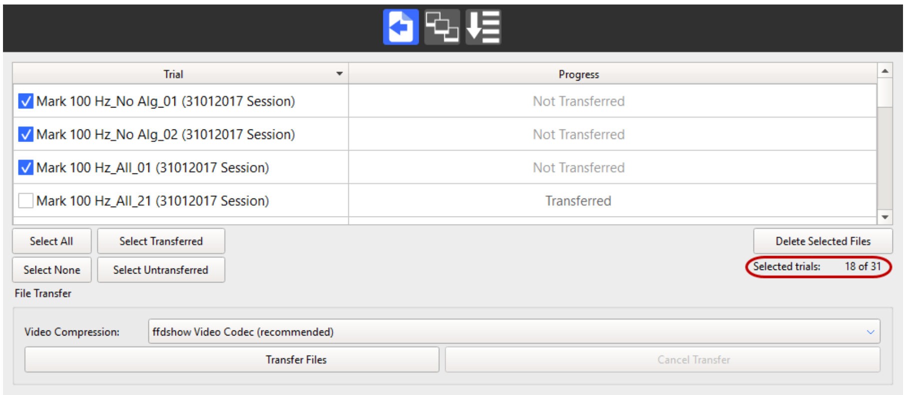
4. Click the **Transfer Files** button to start transferring the video files associated with the selected trials. Any specified transcoding is performed as the first stage of the transfer process.

> **Caution**
>
> Do not attempt to capture data while a file transfer is in process. Interrupting this process may result in problems with the data capture or file transfer. Allow the file transfer process to complete, or click the **Cancel Transfer** button to manually stop the process before starting a new capture.

During the transfer, the progress column in the trials list indicates the transfer status:

- Blank : File transfer process idle
- Yellow moving bar: File transfer process in progress
- Green static bar: File transfer process successfully completed
- Red static bar: File transfer process failed or canceled

In addition, the **File Transfer** counter enables you to check easily on the progress of the transfer of large numbers of video files.



The counter displays both the number of the current video and the total number of videos that are being transferred.

Install the FFDShow codec

The FFDShow codec is recommended by Vicon for transcoding your raw video files.

To install this codec:

1. Visit the Third-Party Plugins page on the Vicon website.
2. Click on **FFDShow Codec** and download the file.
3. Install the downloaded **FFDShow.exe** file.
4. Accept the default options, ensuring that on the **Select Components** screen, the **VFW Interface** option is selected.
5. Click **Start > All Programs > ffdshow > VFW Configuration**.
6. On the **Encoder** tab, ensure **Generic** is selected, then select the following options and click OK:
   - Encoder: **MPEG-4**
   - FOURCC: **XVID**
   - Mode: **one pass - average bitrate**
   - Bitrate (kbps): **900**
   - Maximum I frame interval: **10**
   - Minimum I frame interval: **1**
7. In Nexus, you can now select the FFDShow codec from the list of available codecs in the **File Transfer** area of the **Data Management** tab.

De-interlace AVI files

Nexus can permanently de-interlace all interlaced .avi files associated with the currently loaded trial. The de-interlaced .avi file is placed in the same directory as the source .avi file. It retains the original file name, but is appended as follows: [filename].*interlaced.avi*.

If an .avi file within the currently open trial has been de-interlaced, Nexus automatically loads the new (de-interlaced) version of the file.

> ⚠ **Important**
>
> Audio tracks are not transferred to the de-interlaced file.

To de-interlace an .avi file:

1. To load a digital video file, with the **Camera** view selected, click on the desired movie file on the **Data Management** tab.
2. In the **Available Operations** section of **Pipeline Tools** pane, expand the **System** pipeline operations list.
3. Double-click the **Apply Codec to Video** pipeline operation.
   The operation is added to the current pipeline and is displayed at the bottom of the list in the **Current Pipeline** section.
4. In the **Current Pipeline** section, click on the pipeline operation, and then in the **Properties** section at the bottom of the **Pipeline Tools** pane:
   a. From the drop-down **Video Codec** list, select the codec you want to use for file compression.

      > ⚠ **Important**
      >
      > Due to the rapidly changing nature of available capture hardware, PC processing power, and available codecs, please check the third-party downloads on the Vicon website for the latest recommended codec (currently FFDShow).

   b. If you want Nexus to create a backup of the video file, ensure the **Keep Backup** option is selected.

> **Important**
>
> Reverting to the backup file to re-do the de-interlacing is not supported in Nexus. Therefore, Vicon recommends that **Keep Backup** remains selected whenever you permanently de-interlace an *.avi* file.
>
> To perform de-interlacing again on a video file, you must re-import the file and rerun the pipeline operation.

   c. From the **Remove Interlacing** list, select the required option.

5. Add any other pipeline operations you want to include in the pipeline.

> **Tip**
>
> The pipeline operations are run in the order they appear in the **Current Pipeline** list; you can rearrange the order by dragging operations into the desired position in the list.

6. In the **Current Pipeline** configuration management section, click the Save button to save your settings. (An asterisk (*) is displayed next to the pipeline name if there are unsaved changes.)
7. Run the pipeline.

Work with IMUs

If you are using Vicon IMUs (Inertial Measurement Units) with Nexus, first ensure you have set up the IMUs and Nexus as described in Configure IMUs on page 89.

Important

Vicon IMUs are supported for use with Nexus for research purposes only. For full sensor safety and regulatory details, see the IMeasureU Sensor Safety and Regulatory Information, available from the Vicon website.

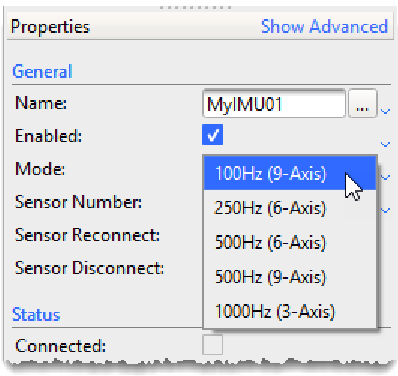
For information on working with IMUs, see the following topics:

- Capture IMU data on page 223
- Manage your captured IMU data on page 225
- Transfer data from IMUs on page 227
- Export IMU data on page 228
- Import data from the IMeasureU Research app on page 229
- Calibrate IMUs on page 231
- Pair an IMU to a different device on page 232

Capture IMU data

1. Having have set up the IMUs to work with Nexus (see Configure IMUs) on page 89, capture in the usual way (see Capture the required movement on page 137).

   While the IMUs are connected to Nexus, a real-time preview of data streaming from the IMUs is displayed. This preview data stream is supplied to Nexus at 50 Hz, regardless of the trial collection rate (100 Hz, 250 Hz, 500 Hz or 1000 Hz). You can view this preview data in Live mode in Nexus.

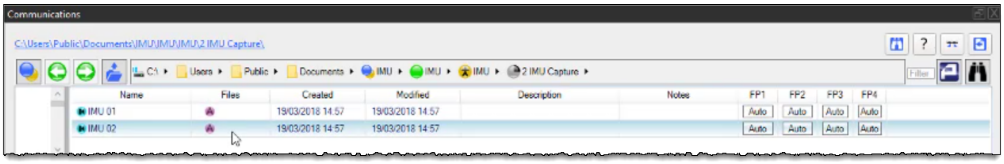


The preview stream is optimized for low latency data visibility and may therefore have small frame gaps where data does not appear.

When trial capture starts, the IMU simultaneously enables the display of preview data at 50 Hz and writes data to its internal memory.

> **Note**
>
> Gaps in the preview data do not indicate gaps in the data that is written to memory.

When you stop a trial capture, an *.x1d* file that contains the preview data that is displayed in the Graph view is saved into the current Session folder and is displayed with a purple A icon on the Data Management tab.



2. When you have finished capturing data, plug the IMUs into your PC using the supplied USB cables.

Manage your captured IMU data

After you have finished capturing data and have plugged the IMUs into the PC, you use the IMeasureU tab in the Nexus Communications pane to work with the data and transfer it.

> **Tip**
>
> To help you decide whether to transfer recorded data, you can check the preview data by playing it in Offline mode first.

To display the IMU data, in the **Communications** pane, click the **IMeasureU** tab.

The connected IMUs and the trials that were recorded onto them are listed.

Depending on what you want to do with the data that you have captured, choose the appropriate option.

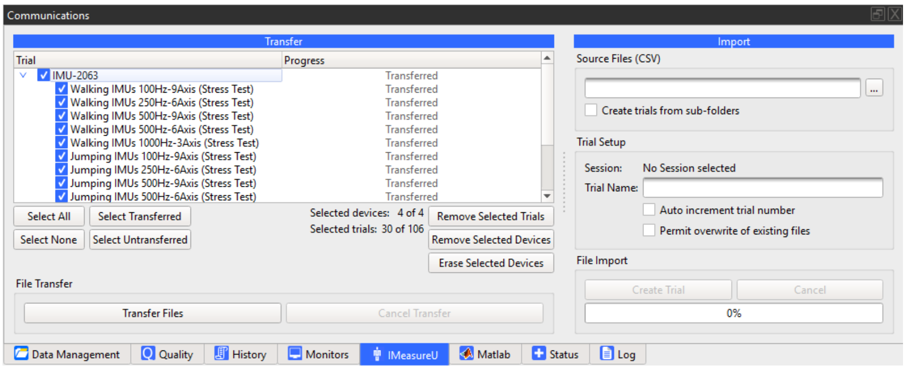
To do this	Take this action
Select one or more trials	Click, SHIFT+click or click **Select All** After you have transferred trials (see Transfer data from IMUs on page 227), you may want to use **Select Transferred**, or **Select Untransferred**.
Remove selected trials from the Transfer list.	Select the trials that you want to remove and click **Re move Selected Trials**. The trials are removed from the Transfer list, but remain on the IMU.
Remove from Nexus all references to selected IMUs and their related trials.	Select the the IMUs that you want to remove and click **Remove Selected Devices**. The selected IMUs and their trials are removed from the Transfer list, but the trials remain on the IMUs.

To do this	Take this action
Remove from one or more IMUs all collected trials, and remove the IMU and its trials from the Transfer list.	Select the the IMU(s) that contains the trials that you want to remove and click **Erase Selected Devices**. All data is removed from the selected IMU(s) and the IMU is removed from the Transfer List.
Transfer trial data from the IMUs to the Session folder in Nexus on your PC.	See Transfer data from IMUs. on page 227

Transfer data from IMUs

To transfer the data from the IMU(s) to the Session folder in Nexus:

1. Ensure you have downloaded and installed the CP210x USB to UART Bridge VCP Drivers.
2. On the IMeasureU tab, in the Transferlist, select the required files for transfer to the Session folder.
3. At the bottom of the pane, click the **Transfer Files** button.
   When transfer is complete, the Progress column in the Transfer list changes to Transferred for the IMUs and for each successfully transferred trial.

   After the transfer operation, for each trial, you can find the following files in the Session folder:

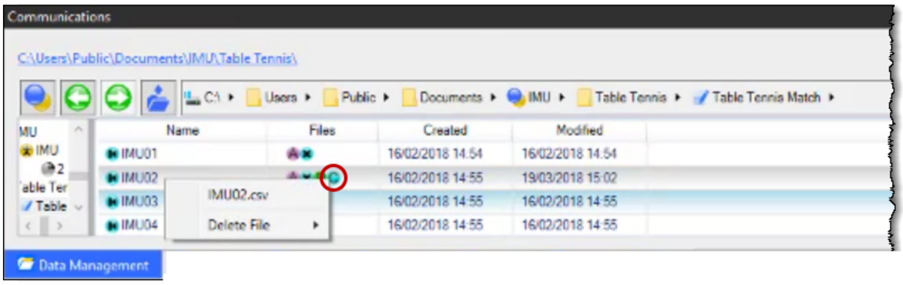
   - An *.x1d* containing the preview data (50 Hz)
   - An *.imu* file containing the higher quality data (100 Hz, 250 Hz, 500 Hz or 1000 Hz) from the IMU
4. You can now view and process the IMU data along with the rest of the trial data.

Export IMU data

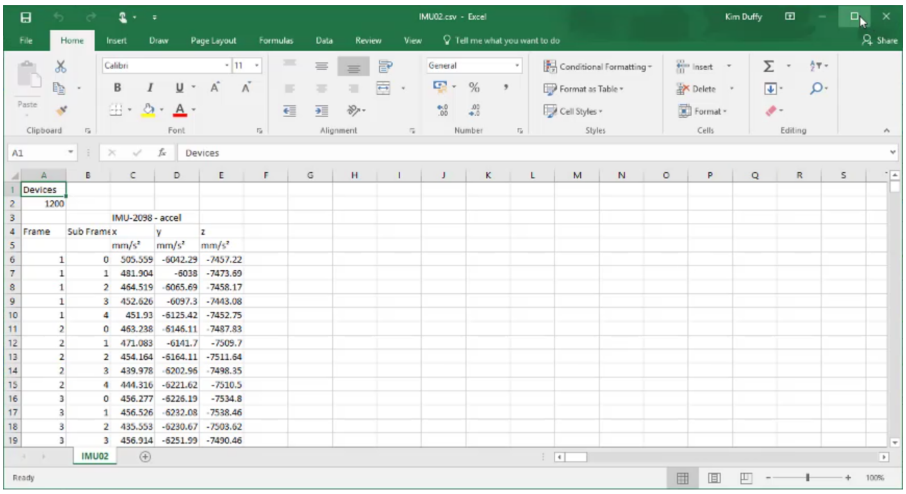
You can export data captured using IMUs by running the appropriate pipeline.

To export IMU data:

1. If you want to select data from only some devices, in the System Resources pane, ensure the required IMUs are selected.
2. In the Pipeline Tools pane, expand **File Export** and double-click **Export ASCII** to add it to the current pipeline.
3. In the Properties pane, ensure that the settings are as required. In the Devices section, if necessary, change the **Devices for Export** list to specify your selection.
4. Run the **Export ASCII** operation.
   To open the exported file in Microsoft® Excel®, on the Data Management tab, click the relevant C icon and then click the filename.



The data is displayed in an Excel spreadsheet.



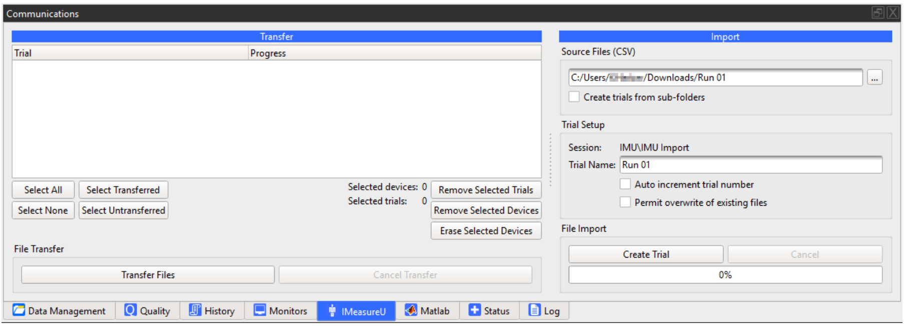
Import data from the IMeasureU Research app

You can capture data using IMeasureU Research app and then import the downloaded CSV files from Lightning into Nexus.

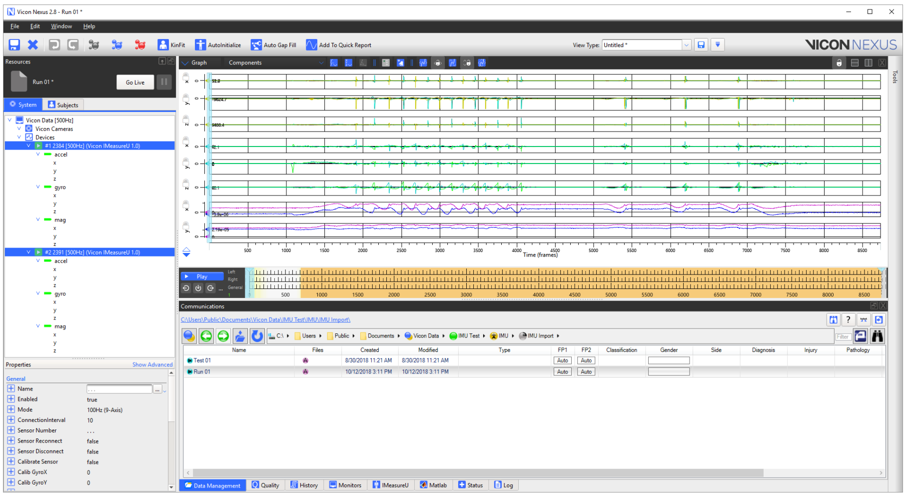
To work with IMU Research input:

1. On the **Data Management** tab, make sure you have selected the session into which you want to import your IMU Research files.
2. On the **IMeasureU** tab, in the **Import** section enter or browse to the relevant folder to import the CSV file(s) that you downloaded from Lightning.

   If you have downloaded multiple files to sub-folders, select **Create trials from sub-folders**.
3. In the **Trial Name** field, enter a name for the trial that will be imported. If required, select the options to automatically increment the trial numbers and/or overwrite existing trials with the same name.



4. Click **Create Trial**.
   Files with the extension *.x1d* are created in the location you selected in Step 1. You can view IMU Research data in the Graph pane in Nexus.



Calibrate IMUs

IMUs are already calibrated when they are supplied to you, and the calibration is stored on the IMU, so you only need to re-calibrate when you update the camera firmware or if an offset of the IMU occurs.

Note that IMUs can be calibrated at 100 Hz only.

To calibrate IMUs:

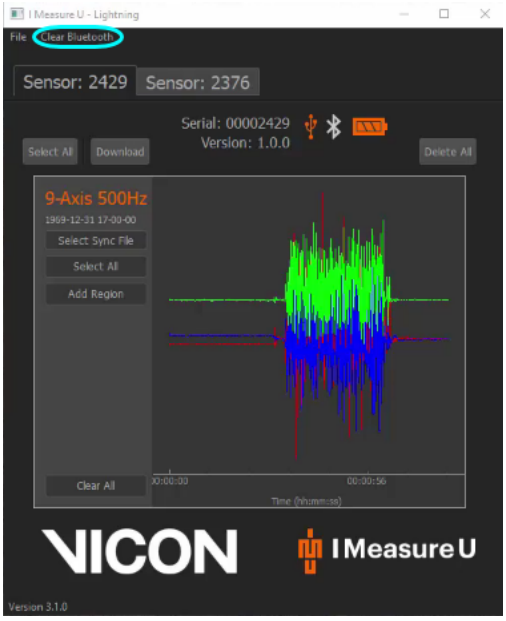
1. Select the IMU(s) that you want to calibrate and in the Properties pane, ensure Advanced properties are displayed.
2. Ensure that the IMUs are static and are not subject to vibration, and positioned with the Z+ axis pointing up or down.
3. In the General section of the Properties pane, click **Calibrate Sensor**.

Pair an IMU to a different device

After an IMU is paired with a Bluetooth® (BLE) dongle (central device), the IMU will only work with that dongle and will reject requests from other central devices (built-in Bluetooth or BLE dongle). To pair the IMU with a different central device (eg a dongle on a different Windows computer), you must erase the stored connection information.

To erase stored connection information:

1. Turn on the IMU and plug it into the PC via USB.
2. Before downloading any data, on the IMeasureU Lightning toolbar, click **Clear Bluetooth.**



This deletes the stored connection information. At the bottom left of the screen a message confirms that the cache has been cleared.

You will not have to repeat a pairing operation unless you:

- Remove or reconfigure your Bluetooth receiver (unplug the dongle etc...)
- Manually remove the pairing

Export trial data

During your work with Vicon Nexus, you can export data for use in other software. Nexus offers a number of options for data export, accessible from the **Available Operations** list in the **Pipeline Tools** pane and from some view panes.

You can also export a video file (.avi) from a 3D or **Camera** view.

For more information, see:

- Configure file export pipeline operations on page 234
- Export 3D workspace as AVI on page 239

Configure file export pipeline operations

You view and change the settings for the supplied export pipeline operations in the **Properties** pane at the bottom of the **Pipeline Tools** pane. For more information, see:

- Configure the Export 3D Overlay Video operation on page 234
- Configure the Export ASCII operation on page 236
- Configure the Export C3D operation on page 238

You can also export to a number of other file formats: to view the available formats, in the **Pipeline Tools** pane, go to the **Available Operations** list and expand **File Export**.

For general information about how to use pipelines, see Work with pipelines on page 131.

Configure the Export 3D Overlay Video operation

You can export the 3D overlay of Vicon optical data over images from a supported calibrated video camera to an .avi file for viewing in other applications using the **Export 3D Overlay Video** operation in the **Pipeline Tools** pane.

With the **Combined View** option in a **Camera** view, you can display Vicon optical data overlaid onto images from a calibrated digital video camera. For example, you can display 3D information, such as the floor grid, markers, and virtual force plates, overlaying the 2D video image. The **Export 3D Overlay Video** operation burns this 3D overlay information into an .avi (digital video file) so that you can view it in other applications, such as Vicon Polygon.

To export 3D overlay information to an .avi file:

1. In the **Camera** view toolbar, from the **View** drop-down list select **Combined** to view Vicon optical data overlaid onto images from a digital video camera.
2. In the **Options** dialog box (F7), select the desired options to configure the visualization of data to suit your needs (for example, you may want only the force vector to appear on the 3D overlay).
3. At the top of the **Options** dialog box, click the Save button to save the configuration you just created. The configuration is saved as an .options file in the appropriate Nexus **Options** folder.

4. In the **Pipeline Tools** pane, create a **File Export** pipeline that includes the **Export 3D Overlay Video** operation. (For a reminder of how to create a pipeline, see Create a pipeline on page 134.)

5. In the **Current Pipeline** operations list, click on the operation, then in the **Properties** section at the bottom of the **Pipeline Tools** pane, view or change settings for the desired properties:

   - From the **View Options Set** list, which is displayed in alphabetical order, select the name of the **.options** file you created in step 3.
   - If required, select an option from the **Video Codec** list. (If you want to de-interlace the .avi file, you must select an appropriate codec.)
   - If the trial is cropped, select the appropriate range of frames to export. If you do not do this, the exported video will be of the whole trial, but will be static in the cropped frames.

6. Run the pipeline either on an individual trial in the **Pipeline Tools** pane or on multiple files using the **Batch Processing** Interface (click **Show File Transfer /Batch Processing interface** and then the **Batch Processing** button on the **Data Management tab**).

About 3D overlay files

The 3D overlay information from each DV camera is stored in a separate file, in the format: **TrialName.DeviceID.overlay.avi** where:

- **TrialName** is the base name of the trial file.
- **DeviceID** is the unique identification number Vicon assigns to a DV camera.
  You can find the **Device ID** in the **System Resources** pane, by expanding the **Video Cameras** node, selecting the desired video camera, and then in the **Properties** section expanding the **Settings** area.
- **overlay** identifies the file contents as the 3D overlay information associated with the video file.
- **avi** is the file extension.

For example, with a video camera with a Device ID of 52883644, if you run the **Export 3D Overlay Video** pipeline operation on a video file named **Walk1.52883644.avi**, the exported 3D overlay file will be called **Walk1.52883644.overlay.avi**.

> **Note**
>
> You cannot open an exported 3D overlay (*.overlay.avi*) file in Nexus.

Configure the Export ASCII operation

The **Export ASCII** pipeline operation enables you to export saved trial data to a plain text file, saved in CSV or TXT format.

To export processed Nexus data to an ASCII file:

1. Ensure you have loaded and processed the required data.
2. In the **Pipeline Tools** pane, create a **File Export** pipeline that includes the **Export ASCII** operation. (For a reminder of how to create a pipeline, see Create a pipeline on page .)
3. In the **Current Pipeline** operations list, click on the operation, then in the **Properties** section at the bottom of the **Pipeline Tools** pane, view or change settings for the desired properties:
   - **Flename:** Do one of the following:
     - Accept the default **Current Trial** setting; or
     - To use a different path and/or filename, click the downward arrow to the right of the box, clear the **Macro** check box, and click the ellipsis (...). You can then enter the required file name, including its extension, for example **my_trial.csv**.
   - **File Extension:** Can be .csv, .txt, or if required, clear the **Macro** check box as described above and then specify the required extension.
   - **First Frame** and **Last Frame:** If required, change these to specify the range that you want to export.
   - **Delimiter:** Do one of the following:
     - Click to choose the delimiter of the exported data, selecting either commas, tabs, or line feeds/carriage returns; or

- To use a different delimiter, click the downward arrow and clear the Macro check box. You can then edit the Delimiter field to specify a combination of ASCII characters (maximum of two characters).

- Local Numeric Format: If you want the exported data to use the local language float number format, select this check box.
- Export Gait Cycle Parameters: To export gait cycle analysis, select this option.
- Export Events: To export events, select this option. If you choose to include events, they are sorted in the output file by type, subject and time of occurrence.
- Digital Device Sampling: Choose the digital devices frame rate and sampling rate options:
  - MX Frames: Exports at the same frame rate as the trajectory data. The exported data may be up-sampled to achieve an integer number of sub-samples per frame.
  - Raw Frames Exports at the original frame rate and sample rate. The exported frame and sub-frame numbers may not correspond to other devices or the trajectory data.
- Local Numeric Format: To export using the local language's float number format, select this option.
- For the rest of the outputs, you can do one of the following:
  - Click to choose to export either none, only the selected output type, or all of them; or
  - To supply a comma-separated list, click the downward arrow to the right of the drop-down list and clear the Macro check box.

> **Tip**
>
> In most cases, as an alternative to selecting All, you can use the asterisk * wildcard.

4. Either run the pipeline or right-click the Export ASCII operation and click Run selected op.
5. After you have run the pipeline operation and exported the file, you can examine the exported data as required.

Configure the Export C3D operation

This pipeline operation exports the current state of the processed data to a .c3d file. You can then import the data into other software for further processing or report generation.

In the **Properties** section you can view or change the following properties:

- **Filename** Name of the file to be exported. By default, this is the name of the current trial, for example **Trial01**. To change the name, click the downward arrow to the right of the field and clear the **Macro** check box. You can then enter a new name.
- **First Frame** First frame of the range to be exported
- **Last Frame** Last frame of the range to be exported
- **Trial Name Postfix** Adds the string you specify to the end of the filename. For example, if you entered export in this field, the name of the output file would be **Trial01.export.c3d**.
- **Integer Format** Measures the maximum range between real data points, and determines a scale factor. The data is then scaled to that range when saved to the c3d file, and all values are written with the Integer format. When the data is read into another program (eg Polygon), the scale factor is applied to the data, converting it into Real data. The Real data format saves the data as it is, without any multiplication by a scale factor, and writes it to the c3d file.
  Certain types of data are best suited for the Real format option because no resolution is given up in the storage of the data. However, bear in mind that not all programs can read both Integer- and Real-formatted c3d files. For more details on the .c3d format, see c3d.org.
- **Subject Prefixes** Prefixes the exported marker labels with the subject's name. (To specify that marker labels are prefixed only when more than one subject's data is exported, choose the Auto option.)
- **X Axis Direction, Y Axis Direction, Z Axis Direction** Enables you to choose the direction of the axis in the exported 3D world.

Export 3D workspace as AVI

The **Export 3D Workspace as AVI** button in the **Camera** view, **3D Perspective** and **3D Orthogonal** views lets you easily create visually rich content for presentations or for use in other third-party applications.

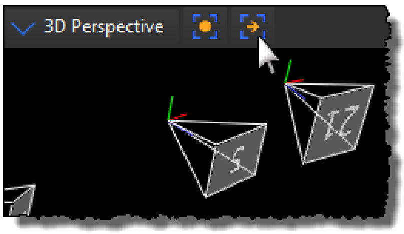
Nexus enables you to create video files of a selected 3D workspace.

Before you export a workspace, ensure you have:

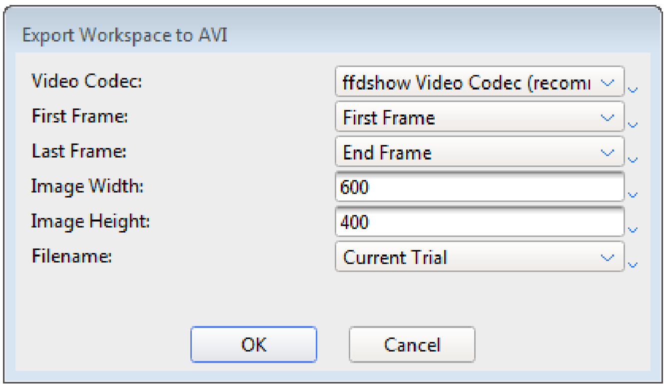
- Loaded a trial.
- Installed the desired video compression codec.

To export a workspace as an AVI:

1. Ensure Nexus is not in Live mode.
2. In the workspace, click the **Export Workspace to AVI** button.



3. In the **Export Workspace to AVI** dialog box, enter the required information and then click OK. Note that **Image Width** and **Image Height** refer to the resolution (in pixels) of the exported image.



A progress bar indicates the status of the export process and by default, a video file with the same name as the current trial is created in the trial session folder. (You can change the name and location if required.)

Further resources

If you need more information than that supplied in the documentation or on the Vicon Support web pages, please contact Vicon:

Denver, CO
Vicon Denver
7388 S. Revere Parkway, Suite 901
Centennial
CO 80112
USA
T: 303.799.8686
F: 303.799.8690
E: support@vicon.com

Oxford, UK
Vicon Oxford
Unit 6, Oxford Industrial Park
Mead Road, Yarnton
Oxford
OX5 1QU
United Kingdom
T: +44.1865.261800
F: +44.1865.240527
E: support@vicon.com

Los Angeles, CA
Vicon LA
3750 S. Robertson Boulevard, Suite 1
Culver City
Los Angeles
CA 90232
USA
T: 310.437.4499
F: 310.388.3200
E: support@vicon.com

Vicon trademarks