6 CT Common Processes

The processes in this section are processes common to CT viewers and applications. Not all the processes are relevant to NM, MR and/or MMV applications.

Series Tree

The series tree is a user interface control that displays a list of the studies and series that are loaded into the viewer or application, and also any other elements (like batches) that have been created.

NOTICE

The Series tree appears in all applications (including the “volumetric” modes of Slab, Volume, and Endo) in nearly identical form.

The series tree’s elements display in a format similar to the way directories, folders, and files are shown in the operating system. Each element of the series tree has a small icon to identify the type of element being listed. These icons include:

 Patient

 Series

 Volume Images

 Batch

 Images

 Image Parameters

 Surview

3000067186691_A/881 * 2021-06-30

Philips

Series-tree Icon Variations

If the element has been saved, its icon is shown in full color. (Batch example shown at left.)

If the element is not currently saved, its icon is shown as faded.

If the element is not loaded, its icon is “broken” (shown with a diagonal white line through it).

If the element is not available, the icon is grayed out.

Series Tree Structure

The tree has three hierarchic levels. Each level acts like a folder.

First Level	Second Level	Third Level
Patient. First Level. 		
	Series. Second Level. 	
	Batch. Second level. 	
		Images. Third Level within Series and Batch. 

300006718691_A/881 * 2021-06-30

First Level	Second Level	Third Level
		Image Parameters. Third Level within Series and Batch. 
		Surview. Third Level within Series and Batch. 

Series Tree Study Folder

The first level is the Study folder and is titled with the Patient Name. It contains all instances related to the Study as seen in the patient directory.

Click on the **Study** folder to select all elements beneath the second level folders but not those in the third level folders.

Click on the **(+)** symbol to expand and the **(-)** symbol to shrink all levels below the Study.

When all the content of the Study folder is selected, the folders get selected. This is also true for unselected.

Series Tree Series Folder

The second level is a folder that contains all instances related to that series and is titled with the Series number and description.

Clicking the series folder selects all elements beneath it but not those in the third level folders.

Click on the **(-)** symbol to shrink all levels below the series.

When all the content of the Series folder is selected, the folders get selected. This is also true for unselected.

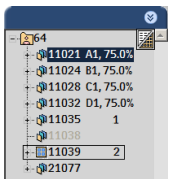
2D Mode

In the 2D mode, under this level appear all the acquisitions: Mixed series are divided into subgroups under the series folder. Surview and Image parameters are identified as such in their names.

Slab, Volume, and Endo Modes

In the volumetric modes (including Slab, Volume, and Endo) the series level represents volume data. Each branch includes a different volume.

Series Tree Batch Folder



A batch appears at the same level as the series and is titled with the batch name. When it is saved, the series number is added before the batch name. Batches are labeled automatically (Batch 1, Batch 2 etc.…), and referenced to the original series.

Series and Batch Folder Contents

Contains three types of acquisitions, which are indicated by three different icons: Images (and, in volumetric viewers, MPRs and volumes); Surview Image; and Parameters.

Duplicate Series

These are added to the tree according to their content, and identified by series number. Duplicated series can be renamed and can be saved.

Series Tree Item States

Temporary (levels 2 and 3). A batch or duplicated series that is not saved in the database. These are identified by a faded icon.

Loaded (levels 1 and 2). Items that are loaded to the memory and can be shown on screen immediately when asked for. These are identified by a full color icon.

Unloaded (levels 1 and 2). Items that require a loading operation when you ask for review. These are identified by a “broken” icon and only appear during the loading of a study.

Select Series Tree Item

Active (levels 2 and 3). The element on the tree that contains the active image in the view port. This is identified by a black background behind the title.

Selected (levels 1, 2 and 3). Determines the data that is available for viewing. These are identified by an azure background under the title. To select more than one element, hold down the <Ctrl> key and click the element(s).

Unavailable (levels 1, 2 and 3). Items are not available to a particular viewer are grayed out.

Work with the Series Tree

A single left click controls the selection.

- A highlighted item remains selected and become active. A non-highlighted item becomes the only one selected (and therefore active).
- Hold down the **<Ctrl>** key and click on an item to select it.
- While holding **<Ctrl>** key down, you can select additional items.
- While holding the **<Ctrl>** key down, click on a selected item to deselect it.

3000067186691_A/881 * 2021-06-30

All included items become highlighted.

A double-left-click on an item makes that item the only one highlighted (and therefore active).

Series Tree Menu Options

Various options and functions become available when you right click on an item in the Series tab.

Launch Application

Right-click a series to display a list of viewers and applications available on your IntelliSpace Portal. Click on the desired selection to launch it with the selected series.

Fuse and Swap

Use the Fuse option to display a fused image of one series displayed as a semi-transparent color overlay on top of the other. The Fuse option should be used after selecting at least 2 series.

After you Fuse the series, the Swap option is available.

Use the Swap option is used to swap between the underlay and the overlay series in fused mode.

Merge

Use Merge to create a single series out of 2 or more series. The new series can be viewed or processed in any application (except Cardiac applications).

For best results, all parameters should be exactly the same. There is some interpolation in creating the new series when there is overlap or missing data, and when parameters are not exactly the same.

Expand and Collapse

This function Is only available on level 1 and level 2 folders.

Click the Expand selection to reveal the contents of the next folder level. (The selection changes to Collapse after you enable Expand.)

Clicking Collapse on a level 1 selection closes all folders, regardless of their selected state.

Clicking Collapse on a level 2 selection closes only the selected folder.

Expand All

Click Expand All to open all folders, regardless of their selected state. Expand All is available on all folder levels.

3000067186 91_A/881 * 2021-06-30

Philips

Remove...

The Remove (data from application) function:

- Removes the entire selected item (highlighted in black) along with the folder from the series tree.
- Removes only the folder's contents, never the Patient folder (level 1).
- Is available on any level.
- Issues a warning message.
- Issues a different warning message if the item has not been saved.

Rename

Rename is only available on unsaved items created in the application.

Series Tree Image Selection

The Selection function is available in the Series tab of the CT Viewer - 2D mode and the Quick Viewer. Use the Selection tools select images. The buttons, from left to right, are:

- One image (at a time).
- A series of images (Select more than one image by holding <Ctrl> and clicking other images, but only one is active. You may select a range of images by holding **<Shift>** and clicking a beginning and ending image.)
- The screen (the whole window).
- All images loaded in the viewer.

Selecting images allows you to perform the same manipulation on the selected images (such as zoom, scroll, pan, swivel and rotate).

When switching between selection modes, the active image in the new mode is the same image that was active beforehand.

NOTICE

Selected images are enclosed within an orange frame.

Images that are not selected have no surrounding frame.

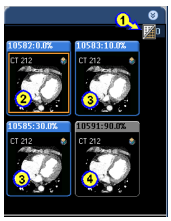
The currently active, selected image is enclosed within a blue frame.

All your image viewing operations are performed in real-time on selected images.

3000067186691_A/881 * 2021-06-30

Philips

View Series as Pictorials



You can view items in the Directory's Series tab either in a list or as pictorials. Click the **View as Pictorial** button (number 1 in the tab image at left) to switch the viewing mode. (Clicking again restores the "View as Table" mode.) The items that were listed in the patient Series tab are shown as mini image pictorials. Highlighting and color coding identify the state of the items:

- (2 in the example) - An orange colored inner frame within a blue frame indicates the series is "active."
- (3) - A blue frame indicates the series is "selected."
- (4) - A gray frame indicates the series is "un-selected."
- A grayed out pictorial is shown when the relevant series cannot be selected within the current work stage.

You can perform the same manipulations on pictorials as you can on the entries in series list.

Hovering the mouse cursor over a pictorial displays a tooltip that provides extended information on the loaded series.

View Directory Series List as Pictorials

You can view items in the Directory's Series tab either in a list or as pictorials. Click the **View as Pictorial**"button to switch the viewing mode. (Clicking again to restores the "View as Table" mode.) The items listed in the patient Series List are shown as mini image pictorials.

Curve Function

The Curve function is available in the CT Viewer's Slab, Volume and Endo viewing modes, but not the 2D viewing mode.

The Curve function allows you to draw a curve for a study (or load an existing curve) to produce a curved path for two purposes:

- To produce a curved path for the Endo viewer's flythrough function.
- To produce a curved Planar Reformat (cMPR) image and the related cross-sectional images.

If no curves exist when you open the Curve dialog, or if the curve(s) for the current study are not loaded, none will be listed in the dialog window.

WARNING

In cases where the orientation annotations are not displayed on the image - you must not assume any specific orientation. For correct orientation information - use only the images which display such information.

3000067186691_A/881 * 2021-06-30

Philips

Existing Curves

If results with saved curves were loaded, they will be listed in the window. Only curves that were planned on the current study are shown in the curve list. When you click in the list on a curve, the curve mode is activated. The curve is displayed in the viewports, and you can edit it, if desired.

Curve Layouts

The available layouts in the curve mode are 1+3 and 2x2.

If you are in the 1+2 layout when you select a Curve, the layout automatically switches to 1+3.

Curve Tools

Use the Curve Tools to activate the curve, draw and edit a curve, as well as view the curve results.

Curve Mode

If a curve has not been created or saved for this study, only the Curve Mode button is available. To activate the Curve mode, click the **Curve Mode** icon.

Draw Curve

The Draw curve icon becomes available, allowing you to draw a curve.

Edit or Move Curve

The Edit/Move icon becomes available, allowing you to edit the control points of a curve or, from the drop-down, move the curve.

Show Curve

Show Curve is active (checked) by default. Uncheck it if you want to hide an existing curve (to make it disappear from the viewports).

3000067186691_A/881 * 2021-06-30

Philips

Show Curve Results Screen

The Curve result screen icon is grayed out if there is no curve drawn or active for the study. The icon is active if a curve exists. When you click the **Curve result screen** icon, the layout changes and shows the cMPR images produced by the curve and the related cross-sectional images along the curve.

Draw Additional Curves

You can draw as many curves as you want. After finishing one curve, click the **Draw Curve** icon to start a new curve. The Curve list adds a new curve name to the list.

Delete or Rename Existing Curve

Right click on a curve name in the curve list. A drop-down menu allows you to select “delete” or “rename.”

Draw Curve - Slab and Volume Views

In the curve drawing mode, you can use any of the view ports to define a curve. Before starting, manipulate the image viewports for best viewing of the anatomy where you will be drawing the curve. Use the middle mouse button or arrow keys to scroll.

1. Click the **Curve mode** icon.

2. Click the **Draw curve** icon (this icon is automatically active if no previous curve exists). The cursor changes to a pencil.

   In the Draw Curve mode, the layout of the display changes:

   – Before you draw, the slab or volume image and 2 reference images are shown. The upper-right viewport is blank.
   – While you are drawing, the cMPR image appears in the upper-right viewport.
   – While you are drawing, the projection of the curve appears on all the reference viewports, and is updated in real time.
   – After you click the Curve results screen, the two cMPR images are shown in the upper-right viewports and the result image is shown in the lower-right viewport (drawing 3, above).

3. Click the location where you want to draw your curve.
4. Point the mouse to the next location and click to place another control point.

3000067186 91_A/881 * 2021-06-30

Philips

5. Repeat the point and click procedure until you are finished drawing the curve.
6. Double click to end the curve.
7. Click on the **Curve results screen** icon to display the cMPR images.

Curves and cMPR Images

Changing the orientation of the images updates the cMPR image.

The cMPR image grows interactively with each click.

The cMPR image can be swivelled around the path.

The curve is shown in all viewports.

Create Curve - Endo Viewer

To create a curve in Endo mode, first do a flythrough, then click the **Create Curve form navigation** icon. The curve is added to list as “Navigation Path”. You can draw the curve in any of the viewports. After you finish drawing (double clicking) in one viewport, you can continue on another viewport by clicking on the end point (the cursor displays a + sign), then clicking on a nearby area.

Edit Curve - Endo

When you activate the Edit Control Points icon you can edit the shape of a curve by dragging its control points. As long as the Curve mode is active, the curve can be edited. You can also add control points and delete points.

Editing can be done in any viewport, including in the Curve Result window.

To have a larger editing screen, you can double-click on any viewport to change it into a 1x1 layout. Double-click again to return to the original layout. Alternately, you can swap the required viewport into the central large viewport by dragging it with the right mouse.

Edit Endo Curve

Active the curve by clicking on it in the list.

Click the **Edit Control Points** icon.

Double click the mouse.

3000067186691_A/881 * 2021-06-30

Philips

Move Curve - Endo

From the drop menu, select **Move Curve**. Drag the curve to the desired location. Double click the mouse.

Add Point to Active Curve - Endo

Move the mouse near the curve. The cursor changes to a plus sign (+) (“add point”). Click on the curve where you want to add a new point. Double-click the left mouse.

Delete Point on Active Curve - Endo

1. Move the mouse over the point you want to delete. Its control point becomes a bold yellow.
2. Hold down the <Ctrl> key. An X appears next to the cursor.
3. Click on the point to delete it; clicking and dragging a control point and releasing it on top of another one also deletes it.
4. Double-click the left mouse.

Extend Curve - Endo

1. Select **Edit Control Points**.
2. Move the cursor off the end until a plus (+) sign appears.
3. Click to place point(s) to extend the curve.
4. Double-click the left mouse.

Alternate Method

1. Select **Edit Control Points**.
2. Right click on the curve.
3. From the menu click **Extend Curve**. A new end point is shown, and you can continue drawing more points.
4. Double-click the left mouse.

Activate Control Points - Endo

1. Place mouse cursor over curve and it turns yellow.
2. Right click.

3000067186691_A/881 * 2021-06-30

Philips

3. Select **Edit Control Points**.
4. Move the cursor off the end until a (+) sign appears.
5. Click to place point(s) to extend the curve.
6. Double-click the left mouse.

Save Curve

To save a curve use the “Save results as...” function in the Common Tools. See section “CT Common Tools” on page 141.

Tissue Segmentation

You can better visualize various tissues of interest by segmenting them with the Tissue Segmentation function.

When you activate the Tissue Segmentation function, all tissue elements that fall within the range of the preset are highlighted with a blue overlay. The current preset Center and Width values are used for the overlay.

Tissue Segmentation Factory Presets (Hounsfield Units)		
	Center	**Width**
Bone	1160	2048
Air	-750	1000
Body	1500	4095

You can create new presets as needed. This can be done here in the Tissue Segmentation function, or in Preferences.

Reference images are automatically assigned the smallest possible thickness, which cannot be adjusted.

When you exit the tissue segmentation function, any unsaved tissue is lost.

On the reference images, tissue that is included in the active tissue is light blue color, the same color as the active tissue in the volume view port.

300006718691_A/881 * 2021-06-30

Philips

The Target Volume, Bounding Cube, or Clipping Planes tools can be used to define the area of interest. See section “Clip & 3D Segmentation Functions” on page 199.

Edit Highlight Presets

Threshold values (highlighting) can be changed by the middle mouse button. Adjusting the threshold does not effect any accumulated pink tissue.

NOTICE

Moving mouse up removes from tissue and moving mouse down adds to tissue.

You can add a preset or edit the preset list in Preferences. See also **Instructions for Use** > **Preferences** > **Segmentation Highlights Presets**.

Save Tissue Segmentation

The **Save As...** function allows you to save tissue segmentation presets that you have created.

1. Create your preset by adjusting the threshold.
2. Click the drop down arrow next to the preset.
3. Select **Save As...**.
4. Enter the preset name in the Define Preset Name dialogbox.
5. Click **OK** to save as a new preset.

NOTICE

The presets that you create and save are saved in **Preferences.**

When you change the threshold value of a selected preset it is first identified as “Modified.” You can then save it as a Preset by renaming it.

Specify Window and Level Threshold (Type In... Function)

The **Type in...** function allows you to specify the Window and Level for the threshold.

1. Click the drop down arrow next to the preset.
2. Click **Type in...** .The Threshold dialog box opens.
3. Click in each box Type in the desired center and width.

   Or:

   Click the up and down arrows to change the values by 50 units for each click.
4. Click **OK** to apply your settings. The dialog box closes.

NOTICE

Apply shows the results but leaves the dialog box open. Cancel closes the dialog box. OK changes the settings to the values you typed in, and closes the box.

Seed

Places a seed to mark a specific volume of Interest (VOI).

WARNING

When placing a seed on the volume image, verify its' location on the reference images.

Anti-seed

Places a seed to remove a specific volume of interest (VOI). Anti-seed removes only those parts that are not connected to the parts that are marked with the "Seed" marker.

WARNING

When placing a seed on the volume image, verify its location on the reference images.

Slice-by-slice Sculpting with Interpolation

To more accurately define a tissue, you can draw individual sculpting ROIs on various slices of a stack of images and the ROIs is interpolated across intervening slices.

WARNING

Verify accuracy of Slice-by-slice-with-interpolation Volume Segmentation and labels (manual visualization). If necessary, correct with the manual correction tools provided by the application.

Within the ROIs, only the data that is marked by the threshold is used in the tissue definition.

You can draw ROIs on as many slices as you want.

300006718691_A/881 * 2021-06-30

Philips

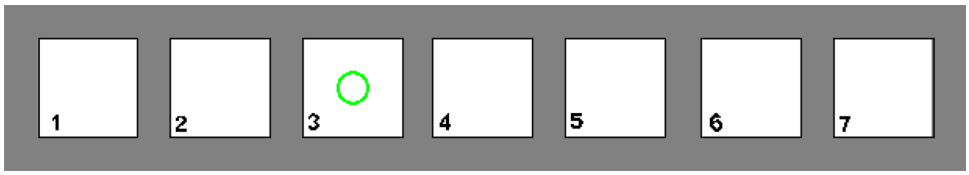
If you draw an ROI on a slice that already has an ROI, the original ROI is overridden.

ROIs that you draw are shown in green.

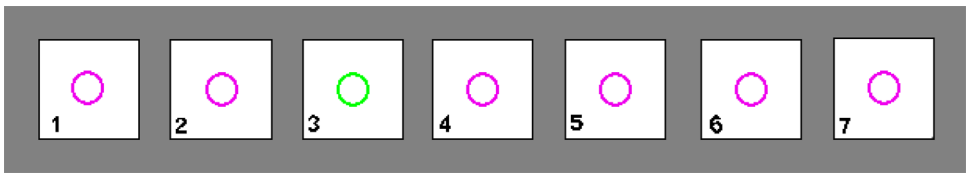
ROIs that the system interpolates are shown in purple.

The diagrams below show how interpolation works:

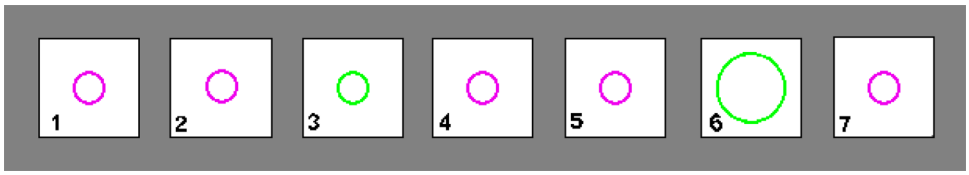
1. Draw the first ROI on, for example, slice 3.



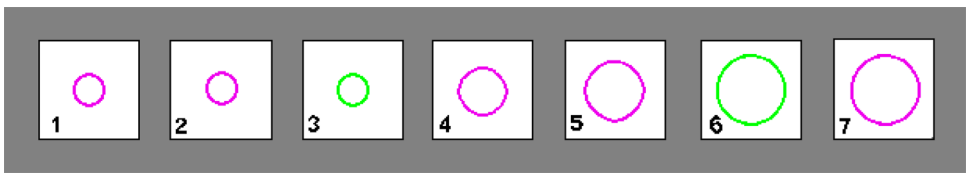
2. The first ROI is projected through the entire stack. A stack of images consists of all images in one of the orthogonal orientations: axial, coronal, and sagittal.



3. Draw another, larger ROI on slice 6.

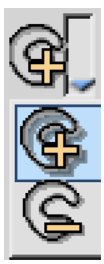


4. The system automatically draws interpolated slices between slices 3 and 6, and the ROI you drew on slice 6 is projected onto slice 7and all following slices.



See also section “Slice-by-slice Sculpting” on page 193.

Slice-by-slice Sculpting



You can sculpt the tissue by adding or removing ROIs on a slice-by-slice basis. Highlight stays on in this mode. These tools are used to make small adjustments. Any tissue segmentation only affects active tissue (blue), not accumulated (pink).

3000067186691_A/881 * 2021-06-30

- **Add Slice by Slice** adds into the current tissue all the blue-highlighted parts within the ROI that you draw.
- **Remove Slice by Slice** removes from the current tissue all the blue highlighted parts within the ROI that you draw.

See also section “Slice-by-slice Sculpting with Interpolation” on page 192.

Add and Subtract from Accumulator

These two tools are used when creating a tissue. They allow you to edit the tissue before you accept it.

1. Create a tissue.
2. Click **Add to accumulator** icon.
3. Make any necessary edits using the sculpting tools. See section “Slice-by-slice Sculpting” on page 193.
4. Click **Add** to or **Subtract** from accumulator.
5. Repeat steps 3 and 4 until tissue is complete.
6. Save the tissue with the **Save Results As...** function in Common Tools. See section “CT Common Tools” on page 141.
7. When the tissue definition is complete, click the **Accept Tissue** icon to save to Tissue Management.

   The original volume comes back and the tissue is stored in Tissue Management.

Fill, Expand, and Erode

These tools are identical to the same tools in the Clip function. See section “Clip & 3D Segmentation Functions” on page 199.

Reset

This function returns the image to the original state.

Undo

The last bone removal, residual, volume tracing or sculpting function that was performed is undone by this function.

3000067186691_A/881 * 2021-06-30

Philips

Brush

Add to unsegmented areas of active tissue by “painting” with the brush.

1. Activate the appropriate tissue segmentation.
2. Select the brush size from the drop down.
3. Drag the mouse across the unsegmented tissue while holding down the mouse button. The brush will “paint” new segmentation in the same color as the selected tissue.

Smart Tissue Segmentation (3D)

Use the **Smart Segmentation Tools (3D)** to improve the 3-dimensional segmentation of various organs and anatomies. In the CT Viewer, click the button to open the floating dialog box that contains the segmentation tool. The dialog box will snap to the active viewport every time it is re-opened (if the box is in the way, drag it to a different location on the screen).

The **Smart Segmentation Tools (3D)** are also available in some CT Analysis applications; however, the tools are accessed via a drop-down on the Functions tabs.

WARNING

Verify the correctness of the volume segmentation and edit as required.

Smart Segmentation (3D) Workflow

WARNING

Verify the correctness of the volume segmentation and edit as required.

1. Click the **Smart segmentation tools (3D)** button.
2. Select either **Draw smart ROI (3D)**, **Draw smart brush (3D)**, or **Draw with auto-centered brush (3D)** (CT Viewer only) from the menu.
3. Draw the region of interest. If necessary, adjust the Adaptiveness and Smoothness in the Parameters dialogue box. See section “Smart Segmentation Parameters” on page 199.
4. Optionally, select one of the Subtract tools to remove tissue from the segmentation.
5. When done, click **Accept tissue** to add the segment to the Tissue Management or the Findings list.

3000067186691_A/881 * 2021-06-30

Philips

NOTICE

For the CT Viewer Slab Stage, 2x2 layout, the volume image does not display the segmented tissue during the tissue segmentation process. To display the segmented tissue on the volume image: accept the tissue; go to Tissue Management tab; click on the volume image. Within the Tissues list un-check the “Volume” option and check the newly created tissue. In volume stage, when segmenting soft tissue, the VR protocol may not be suitable to show the tissue. Use the middle mouse button to adjust the VR protocol to see the tissue in 3D.

For the CT Viewer, when using the segmentation tools the volume rendering image may not display the actually segmented tissue, due to incompatibility between the segmented areas and the active volume rendering preset. For example, in case the selected volume rendering preset emphasizes the contrasted areas and does not show the soft tissues (segmenting a soft tissue area) it will not show up on the volume image. Also, when activating the “Calculate volume” function, it uses the volume displayed in the volume rendering viewport to calculate the volume rather than the actual segmented tissue. To see the segmented tissue properly on the volume image, open the volume rendering presets list and select a preset named: “Opaque.”

Draw with Smart ROI (3D)

Use to semi-automatically create 3-dimensional tissue segments on one of the MPR images.

WARNING

Verify the correctness of the volume segmentation and edit as required.

1. Select the button from the menu.
2. Place the cursor in the middle of the structure you want to segment.
3. Holding down the mouse button, drag the cursor away from the center of the tissue to enlarge. The contour adapts to the edges of the organ or anatomy being segmented. Stop dragging the mouse when the contour hits the edges of the organ or anatomy being segmented.

   While increasing the contour, verify the correctness of the segmentation on all 3 MPR images to ensure the contour is not being over-increased.
4. If necessary, adjust the Adaptiveness and Smoothness in the Parameters dialogue box. See section “Smart Segmentation Parameters” on page 199.
5. To remove tissue from the segmentation, hold down the Alt key and left mouse button while dragging the cursor over the desired area (or use one of the Subtract tools). See section “Subtract Tissue with Smart ROI (3D)” on page 198.

3000067186691_A/881 * 2021-06-30

Philips

6. When done, click **Accept tissue**.

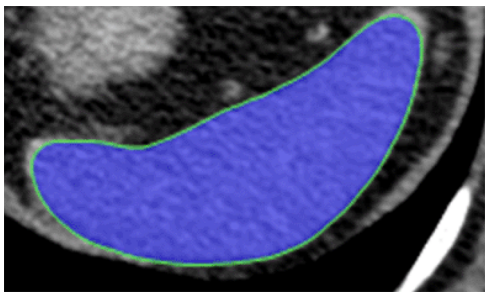
Draw with Smart Brush (3D)

Use to semi-automatically create 3-dimensional tissue segmentation using one or more MPR images. The option is available in the Draw Smart ROI drop-down menu.

WARNING

Verify the correctness of the volume segmentation and edit as required.

1. Select the button from the menu.
2. On one of the MPR images, click the area you want to segment. The green contour appears.
3. To draw a segment using the smart brush, first move the cursor (without pressing the mouse button) to adjust the location of the contour. The contour will follow the cursor.
   - Increase or decrease the contour size using the Ctrl+scroll wheel function. Optionally, use the bracket keys “[” and “]” to change the contour size.
   - When the contour has selected the appropriate tissue, click the image again to add the bounded area to the segmented tissue.



4. To draw a segment using the continuous-draw mode, hold down the mouse button and drag the cursor over the desired tissue. When done, release the mouse button. The marked parts are added to the segmented volume.
   - Optionally, hold down the Shift key while dragging the cursor to auto-center the brush on the highlighted tissue. Auto-center helps find the center of the current tissue and scrolls the dataset. This may be useful when identifying vessels and other prolonged tubular structures.
5. If necessary, adjust the Adaptiveness and Smoothness in the Parameters dialogue box. See section “Smart Segmentation Parameters” on page 199.
6. To remove tissue from the segmentation, hold down the Alt key, move the cursor over the desired area, and click the mouse button (or use one of the Subtract tools). See section “Subtract Tissue with Smart Brush (3D)” on page 198.

3000067186691_A/881 * 2021-06-30

Philips

7. When done, click **Accept tissue**.

Draw with Auto-center Brush (3D)

Use to semi-automatically create 3-dimensional tissue segments on one of the MPR images. The Draw with auto-center brush tool is optimized to help identify vessels and other tubular structures. When using the tool, follow the instructions given in the "Draw with Smart Brush" section. See section “Draw with Smart Brush (3D)” on page 197.

The only difference is that the tool automatically identifies the center of the current tissue and scrolls through the dataset (this is the same functionality as holding down the Shift key while using the Smart brush continuous-draw mode).

WARNING

Verify the correctness of the volume segmentation and edit as required.

Subtract Tissue with Smart ROI (3D)

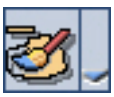
Use to remove tissue segments on one of the MPR images. When using the tool, follow the instructions given in the "Draw Smart ROI" section (see section “Draw with Smart ROI (3D)” on page 196). However, the tool removes (instead of adds) tissue to the segment.

WARNING

Verify the correctness of the volume segmentation and edit as required.

To add tissue to the segmentation, hold down the Alt key, move the cursor over the desired area, and click the mouse button.

Subtract Tissue with Smart Brush (3D)



Use to remove tissue segments on one of the MPR images. When using the tool, follow the instructions given in the "Draw with Smart Brush" section (see section “Draw with Smart Brush (3D)” on page 197). However, the tool removes (instead of adds) tissue to the segment.

WARNING

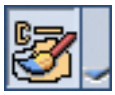
Verify the correctness of the volume segmentation and edit as required.

3000067186691_A/881 * 2021-06-30

Philips

To add tissue to the segmentation, hold down the Alt key, move the cursor over the desired area, and click the mouse button.

Subtract Tissue with Auto-center Brush (3D)



Use to remove tissue segments on one of the MPR images. When using the tool, follow the instructions given in the "Draw with Auto-center Brush" section (see section “Draw with Auto-center Brush (3D)” on page 198). However, the tool removes (instead of adds) tissue to the segment.

WARNING

Verify the correctness of the volume segmentation and edit as required.

To add tissue to the segmentation, hold down the Alt key, move the cursor over the desired area, and click the mouse button.

Smart Segmentation Parameters

Use the Parameters dialogue box to control the tool’s segmentation borders and adaptiveness to curved edges during selection.

- Use the Adaptiveness to control how closely the segmentation adapts to curved edges and borders of the tissue. For round tissues with relatively smooth edges, a setting lower than the default value may be helpful.
- Use the Smoothness to adjust the segmentation borders. At its highest setting the smoothness is a perfect circle. The smoother the segmentation the less sensitivity to noise the selection is. When selecting non-smooth tissues, it may be necessary to use a lower smoothness setting.

The default parameters are used every time you launch a new segmentation tool. If necessary, adjust the parameters for each individual case. The last used parameters will be shown next time a tool is used within the same application session.

Clip & 3D Segmentation Functions

The Clip & 3D Segmentation tab offers a group of functions that you can use to isolate (segment) tissues for better viewing of volumes of interest.

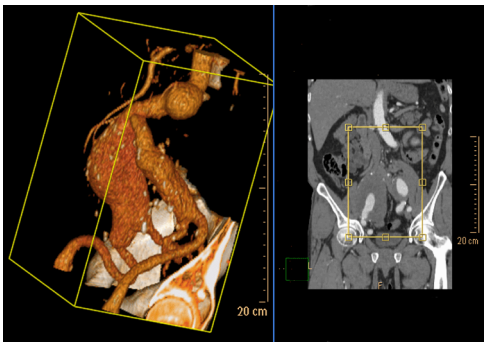
Clipping functions include Target Volume, Bounding Cube, Segmentation Tools, Clipping Plane, etc. Only one of these functions can be used at a time.

Target Volume

Target Volume is used for segmenting large, complex volumes, like the heart and aorta.

3000067186691_A/881 * 2021-06-30

Philips



1. Select the **Target Volume** tool. The target volume box appears in all the viewports.

2. Move the cursor over a line of any box of a reference image. Control points appear on the box.
3. To move the box, place the cursor between the control points. The cursor turns into a cross. Left click and drag the cursor to move the box to the correct position.
4. To resize the box, place the cursor over one of the control points. Left click and drag the cursor to resize the box.
5. To rotate the clipped volume (with the box), use the Swivel function on the volume image.
6. To rotate only the volume, select **Lock Volume**. Now when you drag the volume, the box remains stationary.

OR:

To rotate the plane, select **Lock Plane on Screen** from the drop-down (if available). Now when you drag the plane, the volume remains stationary.

7. To hide the lines in the volume viewport that define the target volume, click the **Hide/Show** icon. (The lines remain in the reference images.) Click the **Hide/Show** icon again to make the lines reappear.

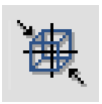
8. To reset the target volume box to the original size when the mode was activated, click the **Reset** icon.

3000067186691_A/881 * 2021-06-30

Philips

9. Click the **Target Volume** icon again to exit the mode. The box disappears and the entire volume image reappears.

Bounding Cube



The Bounding cube is used for analyzing small objects, like neuro vessels. It works like Target Volume and provides a very quick method for focusing in on an object.

When you turn on the cube, a crosshair appears on the reference images along with a yellow cube.

The crosshairs and the cube are active graphics. By moving any crosshair, you move the cube. By moving any edge of any square you change the cube's total dimensions (it always remains a perfect cube).

1. Select **Bounding cube**.
2. To move the bounding cube, point to the intersection of any crosshairs and drag with the mouse, or move the mouse between control points.

   Or:

   Place the cursor between two control points on the cube. The cursor turns into a cross. Left click and drag the cube to the correct position.
3. To change the size of the cube, move the cursor over a line of any cube. Control points appear on the cube.
4. To expand or contract the cube in all dimensions around the cube’s center, use the middle mouse wheel over the volume image. (Alternately, you can adjust the control points to change the cube size.)
5. Use the Lock/Unlock, Hide/Show, and Reset icons. See section “Target Volume” on page 199.
6. When you lock the bounding cube and swivel the volume, the reference lines disappear in the reference viewports. The reference lines reappear when you reset the bounding cube.
7. Click the **Bounding Cube** icon again to exit the mode. The cube disappears and the entire volume image reappears.

NOTICE

Part of the bounding cube can be placed outside the volume. Scrolling up increases the cube’s dimensions; scrolling down decreases them.

3000067186691_A/881 * 2021-06-30

Clipping Plane

The Clipping Plane is a single, movable, infinite plane that cuts through (“slices”) the true volume. The clipping plane removes the volume on the one side of the plane and leaves a volumetric view on the other side of the plane, showing a cross-section of the anatomy at the plane.

1. Click **Clipping Plane** to activate the function. The clipping plane is defined by a line on the reference images. The center of the plane is a straight line that appears on the reference images. No clipping plane lines are visible in the volume view port. The default orientation of the clipping plane is perpendicular to your display screen.
2. To change the cut part of the volume:
   - Rotate the yellow line on the reference images by placing the cursor on either end of the line until the cursor turns into a rotating arrow.
   - Swivel the volume image using the Lock mode.

   By default, when you activate the Clipping Plane mode, the Lock volume icon is ON. This locks the volume, so the clipping plane rotates but the volume does not rotate.

   When Lock volume is OFF, the volume swivels. You can access the drop-down menu to Lock plane on screen. In this mode, the volume can be swivelled while the plane is locked in position parallel to the screen.
3. The Reset icon resets the clipping plane back to its original position when the mode was activated.

   
4. Click the Clipping Plane icon again to exit the mode. The plane disappears and the entire volume image reappears.

NOTICE

Scrolling up moves the plane perpendicular to itself, posterior to the anatomy. Scrolling down moves the plane perpendicular to itself, anterior to the anatomy.

Bone Removal (Create Tissue)

WARNING

Verify bone removal does not affect vessel completeness.

3000067186691_A/881 * 2021-06-30

Philips

NOTICE

Bone removal is intended for use with the body, not the head.

Bone Removal Threshold (3D)

Bone removal is a threshold-based tool. It defaults to 350HU (Hounsfield Units). Bone Removal affects connectivity between objects. When you click on a point, all areas connected to the object that have a HU value greater than the threshold are removed. The extent of the removal function depends also on the volume rendering protocol.

NOTICE

The size and shape of anatomies can change when tweaking a protocol. To prevent wrong interpretation, follow the recommended protocol modification procedure in the application you are using.

(3D) Bone Removal Procedure

1. Click the **Place Seed (3D)** icon. The cursor turns into a pencil.

2. Set the threshold for bone removal (or leave at the default setting).
3. Click on the bone tissue on any image. After a few moments the tissue is removed.

WARNING

When placing the seed on the volume image, verify it's location on the reference images.

4. Continue in this manner to remove all the bone tissue.

   You can fine tune bone removal by adjusting the threshold as you work. If some bones were not removed, lower the threshold and try again. If too much was removed, raise the threshold and try again.
5. If bone removal does not remove smaller, unattached volumes completely, it may be helpful to click the **Remove residuals** icon.

3000067186691_A/881 * 2021-06-30

Philips

WARNING

After bone removal, verify that other anatomical features were not mistakenly removed. Verify Remove residuals does not affect vessel completeness.

6. Save the tissue with the “Save Results As...” function in Common Tools.

See section “CT Common Tools” on page 141.

7. When the tissue definition is complete click **Accept Tissue**. (The original volume comes back and the tissue is stored in Tissue Management).

The Undo/Redo function allows you to reverse your most recent action, in this case, removing bone and residuals. (Undo also works on volume tracing or sculpting. Redo allows you to reverse the Undo.

Use **Reset** to return the image to the original state.

Remove Bone Residuals (3D)

If bone removal does not remove smaller, unattached volumes completely, the Remove residuals tool may be helpful. Residual bone volumes are usually around 20 to 30cc - the system defaults to 20cc. (The cc parameter is used because the size of the fragments that may remain should be small.) All residuals smaller than the specified volume will be removed.

Volume Tracing (3D)

The Volume Tracing function helps you manually segment one or more anatomic structures within the volume image.

WARNING

Verify correct semi-automatic volume segmentation for bed, head holder, Bone and Skull removal operations, and Volume segmentation in volume tracing.

When using the 3D Dye Injection tool verify the correctness of volume segmentation. If necessary, correct the dye tracing using correction tools supplied by this application.

3000067186691_A/881 * 2021-06-30

Philips

Volume Tracing mimics the effect of a syringe that injects “dye.” The dye spreads in the volume, according to the CT value of the surrounding areas. This creates a tissue that can be excluded or isolated from the volume.

Inject Dye (3D)

The Inject dye function is used on the reference images to create a tissue of the volume of interest.

NOTICE

In the CT Viewer, click the button to open the floating dialog box that contains the **Inject Dye (3D)** tool. The dialog box will snap to the active viewport every time it is re-opened (if the box is in the way, drag it to a different location on the screen).

Injection Parameters

You can vary the Injection Parameters as desired to control the injection.

Rate

The Rate parameter adjusts the cubic volume of the injection. The Slow dose is 400 cc; Medium dose is 1500 cc; Fast dose is 4,000 cc. You can also type in an injection rate between 1 and 10,000 cc.

Type

The Type parameter adjusts the “viscosity” of the injection. Injection is fastest at the 1 value and slowest at the 10 value.

Injection Procedure

WARNING

Verify segmentation correctness. If necessary, correct tracing with correction tools.

1. Manipulate any of the reference images to locate the volume of interest.
2. Alter the Rate and Type parameters if desired. See section “Injection Parameters” on page 205.
3. Click the **Inject Dye (3D)** icon (contrast) to activate the segmentation (injection) function.

3000067186 91_A/881 * 2021-06-30

4. Point the mouse onto the volume of interest in the reference image. You cannot use volume tracing on the Volume image.
5. To inject the contrast, click and hold the left mouse button. The volume of interest fills. The injected contrast is displayed as a blue overlay, filling the volume in real time.
6. You can speed up the injection by dragging the mouse to locations within the volume that are unfilled.
7. Release the left mouse button to stop the injection.
8. Wait a moment, then view the results in the volume image viewport.
9. You may continue injecting as necessary.
10. Scroll the images to view more of the volume and continue injecting if necessary.
11. When you are finished, verify the segmentation correctness.
12. Save the tissue with the “Save Results As...” function in Common Tools.

See section “CT Common Tools” on page 141.

13. Click the Accept Tissue icon to save to Tissue Management.

NOTICE

The entire volume image is displayed after you Accept tissue. You can then begin another Volume Tracing session on a different volume of interest.

Fill

The Fill function adds to the injected soft tissue, filling in any holes within the volume. The holes are filled as long as the button is pressed.

NOTICE

In some applications, the Fill button is replaced by the "Fill holes" checkbox. Check the box to fill in the holes. Deselect the box to show the holes.

300006718691_A/881 * 2021-06-30

Philips

Expand (3D)

The Expand function allows you to increase the edges of the tissue. Each click expands the edge by a one-voxel increment.

Erode (3D)

The Erode function allows you to decrease the edges of the tissue. Each click reduces the edge by a one-voxel increment.

Undo/Redo

The Undo/Redo function allows you to reverse your most recent action. If, for example, you over-inject, you can correct the result with the Undo function. Each click of Undo erases the last voxels that were added to the tissue during Inject, Fill, Expand, or Erode. Redo allows you to reverse the Undo.

Eraser (3D)

The Eraser function allows you to remove the contrast from reference images by hovering over the contrast and clicking the left mouse button (you can also hold down the button and drag). The Eraser is a sphere whose radius you can set. The Small eraser is 5 pixels; Medium is 10 pixels; Large is 30 pixels.

NOTICE

Because the eraser is the shape of a sphere, it erases from the volume, not only the slice you use it on. Be sure to verify the results of the eraser by scrolling the reference images.

Show/Hide Volume

Show(/hide) volume turns on and off the original volume image in the main display. Turning off the original volume leaves only the volume that was segmented by the injection process.

Show/Hide Injection Overlay

Show(/hide) injection overlay turns the blue overlay on and off in the reference images.

Volume Sculpting (3D)

Manual sculpting tools are available for volume removal. Three Region of Interest (ROI) sculpting tools are available: Freehand, Rectangle, and Circle. The Exclude function removes everything enclosed within the ROI. The Include function removes everything outside the ROI.

1. Select an ROI sculpting tool. The cursor turns into a pencil.
2. Outline the volume you want to include or exclude from the tissue on a volume image, so that a boundary is formed around the tissue.

   If the resulting Volume of Interest does not provide sufficient separation of the tissue from its surroundings, use the Undo function and then re-draw the ROI.
3. Save the tissue with the “Save Results As...” function in Common Tools.

   See section “CT Common Tools” on page 141.
4. Click the Accept Tissue icon to save to Tissue Management.

Batch Functions

A batch is a sequential set of patient images obtained from the original study or from images processed in a viewer or an analysis application. You determine the composition of the batch by performing the desired image preparation functions and then specifying the starting and ending images of the batch. A batch can be saved as a Windows movie file for viewing on a personal computer.

2D Batch

You can batch a sequence of 2D (original) images from part of a study. This allows you to exclude anatomy that is not in your region of interest, or to skip every two or more intervening images while batching the sequence. This results in a smaller study, which saves on processing and storage requirements.

3000067186691_A/881 * 2021-06-30

Philips

Slab Batch

You can batch a sequence of Slab images, whose thickness and increment spacing you can choose, based on your viewing requirements.

Volume Batch

You can batch a sequence of volume rendered images. (This batch type is typically used in Analysis applications.) A volume batch is often used for viewing a study in the cine mode.

Endo Batch

You can batch a sequence of virtual Endoscopic images. An Endo batch consists of volume rendered images created while following a path within an air, vessel, or contrasted structure. The Endo batch also is often used for cine viewing and movie creation.

When to Use Batch Functions

Batch is always available in all CT Viewers and all CT Analysis applications. Each of the CT Viewers has a batch capability specifically designed for that viewer. Each of the Analysis applications has a batch function that utilizes one (or more) of the CT Viewer batch types. (Different batch types may be offered, depending on what stage of work you are in within the application.)

Create Multiple Batches for Same Study

There is no limit on the number of batches you can make for a study. For example, you can activate a study's sagittal reference viewport, and make and save a batch of sagittal images. Then, you can activate the coronal viewport and make and save another batch.

Save Batch

A batch can be saved in DICOM format, which can then be manipulated the same as any other study.

A batch can be saved as a Secondary Capture (also DICOM format), which makes it more readable by other imaging systems.

A batch can be saved as standard graphic images (JPEG, TIF and BMP), which allows it to be included in many applications, such as word processors, Web sites, personal computers, etc.

A batch can be saved as a movie (AVI or WMV), which is useful in group presentations, on the internet, and by the public media.

Film Batch

A batch can be sent to the Film application of the IntelliSpace Portal.

Send Batch to Report

A batch can be sent to the Report application.

Define Batch Options

The best process for defining a batch depends on your needs, on how you use IntelliSpace Portal, as well as on the individual application you are using.

Mark First and Last Images

This is the basic form of batch definition. Using the mouse in the viewport of your choice, scroll the images in the viewport until you see the first image for your batch. Click the **Mark First** icon. Now scroll to the last image and click **Mark Last**. The batch is complete, ready to be viewed, saved, filmed, etc.

Enter First and Last Image Numbers

In the 2D batch mode, you can use the keyboard to type in the numbers of the first and last images of the batch.

Select All Images

In the Slab batch mode you can click a button to select every patient image for creating the batch. (Also, from the reference image viewport of the Volume batch mode, you can select the All images parameter.)

Select Specific Images

In the 2D batch mode, you can use the “Select every” function to indicate how many images to skip between the each image you want to include in the batch. (First, you should mark the first and last images of your intended batch.)

Mark Individual Images

This feature is used to make a unique (one-time) batch for movies. Here’s how it works: In the Volume batch mode, after finding and marking the first volume image of your proposed movie, you can navigate to any number of successive images and click **Mark Last** each time. (Between each navigation to the next image you can change viewing parameters such as zoom, pan, window/center, swivel, etc.). When you are finished, the batch will consist of a sequence of images for which you clicked Mark Last. (The sequence is interpolated, so that the result is a smooth transition through the images.)

Use Plan on Surview (2D Mode Only)

If a surview exists for the study and is loaded, you can create a batch directly from the surview image. If surview is not loaded you can create a batch from an automatic reference image you can plan from. Define the batch by dragging the surview’s blue overlay to the beginning and end slices of the desired batch.

3000067186 91_A/881 * 2021-06-30

Use Quick Batch

When using a volume viewing mode you can, with a single click of the **Quick Batch** button, create a rotating batch of the entire volume. (If desired, you can adjust the extent of the rotation and the number of images).

Use Batch Preset

If you regularly use batches during post processing, you can define, save and re-use your own batch preset protocols for automatic batch creation. (Clinical Administrators can create batch presets to Share with all users, which are available to all existing and new users of the originating IntelliSpace Portal system.)

Create Batch from Curve

In the Slab, Volume, and Endo viewing modes you can draw curves which can be used to create batches (useful for making Endo movies).

Create Batch from Endo Flythrough

As you fly through an Endo volume, the navigation path is temporarily saved. You can activate the curve and use it to create an Endo batch, which can be saved as a movie.

Create Time-mode Batch

A Time Mode Batch is available in Comprehensive Cardiac Analysis, Cardiac Viewer, Brain Perfusion and Functional CT applications. In Comprehensive Cardiac and Cardiac Viewer this batch mode creates a beating heart movie showing the same anatomical location changing along the various cardiac phases.

In Brain Perfusion and Functional CT applications the time mode batch shows the same anatomical location changing in time.

The Time Mode Batch can be activated by selecting the "Time" checkbox in the "Batch" tab.

Add Additional Images to Batch

You can add these extra frames of information to a batch: mini images, reference images, and patient study parameters.

Batch Preview

Preview is a static display of the batch images, allowing you to preview the created batch before saving it. You can scroll to view all images, and if satisfactory, save, film, and/or send them to the Report function. The “Add to series tree” check box also appears, which allows you to save images the current series list.

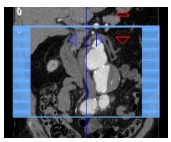
3000067186 91_A/881 * 2021-06-30

To see a preview of the batch you have created, click the **Preview** icon. The Preview function is the same in all viewing modes. For more information, see section “Preview 2D Batch” on page 216.

Batch Cine

Click the **Cine** icon in Common Tools to view the batch in cine mode. You can play the batch backwards and forwards, and speed up or slow down the display. See section “View Batch in Cine (Automatic Scrolling)” on page 225.

Clear Batch



You can discard the batch (without saving) at any time with the Clear icon. You can then start a new batch. After saving a satisfactory batch, click Clear and you can create another batch (as often as desired).

Blue lines at top and bottom show the extent of the batch. Blue tick lines along the left and right edges mark the locations of the individual batch images. The parameters frame is placed first in the batch and the reference images are placed last.

Create 2D Batch

The batch function in the 2D viewing mode allows you to batch a sequence of 2D (original) images for saving, reporting, cine and filming purposes.

In addition to the usual functions provided by batch, the 2D batch mode has 2 main features:

- You can skip every other (or more) intervening images while creating a batch. This results in a smaller study, which saves on processing and storage requirements.
- You can create a batch from a Plan on Surview, manipulating the Surview’s blue overlay to define the batch.

You can define a batch in these ways:

- use the first and last image icons;
- type in the slice numbers; and
- use the up/down arrows.

Plan Batch on Surview - 2D Batch

In the Plan Batch on Surview mode, you can define a batch in these ways:

- type in the slice numbers;
- drag the top and bottom lines in the blue overlay on the surview; and
- scroll the two reference images to identify the start and end locations desired.

If more than one series is in the study, left and right arrows appear, so you can access another series. With the previous and next series arrows, you can page through the series and set batches for each series.

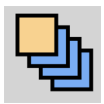
3000067186 91_A/881 * 2021-06-30

Philips

Use Image Mode to Create 2D Batch

To create a batch you can click on the first and last images, or you can enter the image numbers of the images you want to be first and last.

1. Scroll to and select the image you want to be the first image of the batch.
2. Click **Select first image**. The image number appears in the number box.



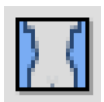
3. After the first image has been designated, scroll to and select the image you want to be the last image of the batch.
4. Click **Select last image**. The image number appears in the number box.

NOTICE

You can also click inside the first image number box and type in the number of the first image. Then, click inside the last image number box and type in the number of the last image.

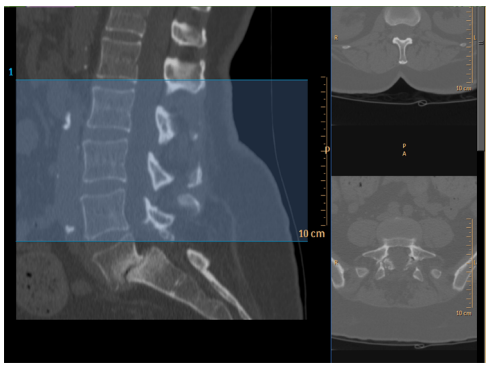
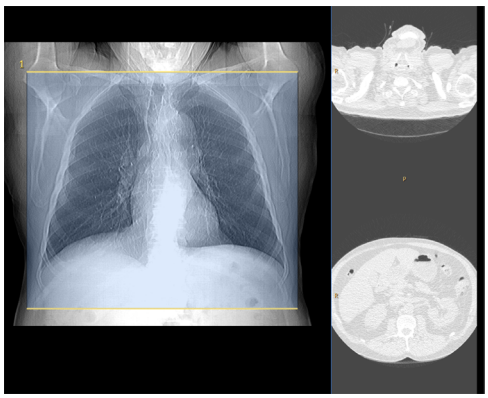
Use this icon to clear the current defined batch series. You can start defining a new batch if desired.

Use Plan on Surview to Create 2D Batch



You can define a batch with the Plan on Surview function using the Surview image. You can define a batch using this function even if there is no surview available. When there is no surview, the system uses a sagittal or coronal reference image. The default is the sagittal image - to change to coronal, roll the middle mouse button.

Click **Plan on Surview** in the Batch tab. The Plan on Surview window displays, as shown below.

Without Surview Image	With Surview Image
	

The main viewport shows a surview image with a blue graphic box. Instead of the Select First / Select Last method of defining the batch, you can use the blue box to define the surview batch.

The top of box is the first slice in the batch (labeled with a 1) and the bottom of the box is the last slice in the batch.

Drag Top and Bottom Lines to Define Surview Batch

With the mouse pointer, approach the solid green line at the top or bottom of the blue box. The line turns yellow.

Drag either or both lines to the desired location(s).

The slice numbers of the batch you have defined appear in the First/Last number boxes.

When working with Plan on Surview options, you must save the batch before moving to the next series.

Drag Entire Box to Define Surview Batch

Point inside the body of the blue box. A cross cursor appears.

Drag the box up or down to the desired location.

The slice numbers of the batch you have defined appear in the First/Last number boxes.

When working with Plan on Surview options, you must save the batch before moving to the next series.

Scroll Reference Viewports to Define Surview Batch

Point inside the reference viewport(s).

Drag (scroll) the reference image(s) to the desired start and end images of the desired batch. The blue box in the main image updates.

The slice numbers of the batch you have defined appear in the First/Last number boxes.

When working with Plan on Surview options, you must save the batch before moving to the next series.

3000067186691_A/881 * 2021-06-30

Philips

Select Every - 2D Batch

You can reduce the total number of images in a 2D batch with the Select every function (select every one, every second, every third, …).

To skip images between the first and last selections:

- Use the up/down arrow buttons to change the Select every setting from the default (select every 1) to any number between 2 and the last number of the series.

  Or:

- Type a number in the box. For example, if you select 3, only every third image becomes part of the batch (image 1, image 4, image 7, etc.).

Add Other Images to 2D Batch

You can add other images like a Parameter frame and a Reference image to the batch.

Add Parameter Frame

Click the icon to add an image of the series (study) parameters to the Batch.

Add Reference Image

Click the icon to add a reference image to the Batch. The Reference Images window is opened.

NOTICE

The Image Parameters page always appears at the beginning of the batch and the Reference images always appear at the end of the batch.

Save 2D Batch as Permanent Series

To create a permanent series from a batch:

- Define the batch using the Image mode option, or the Plan on Surview option.
- Click on **Save Batch As...** The batch is saved as a new series.

Save 2D Batch as Temporary Series

To create a temporary series from a batch:

- Define the batch using the Image mode option.
- Click on **Duplicate** in the upper toolbox. A new temporary series is created.

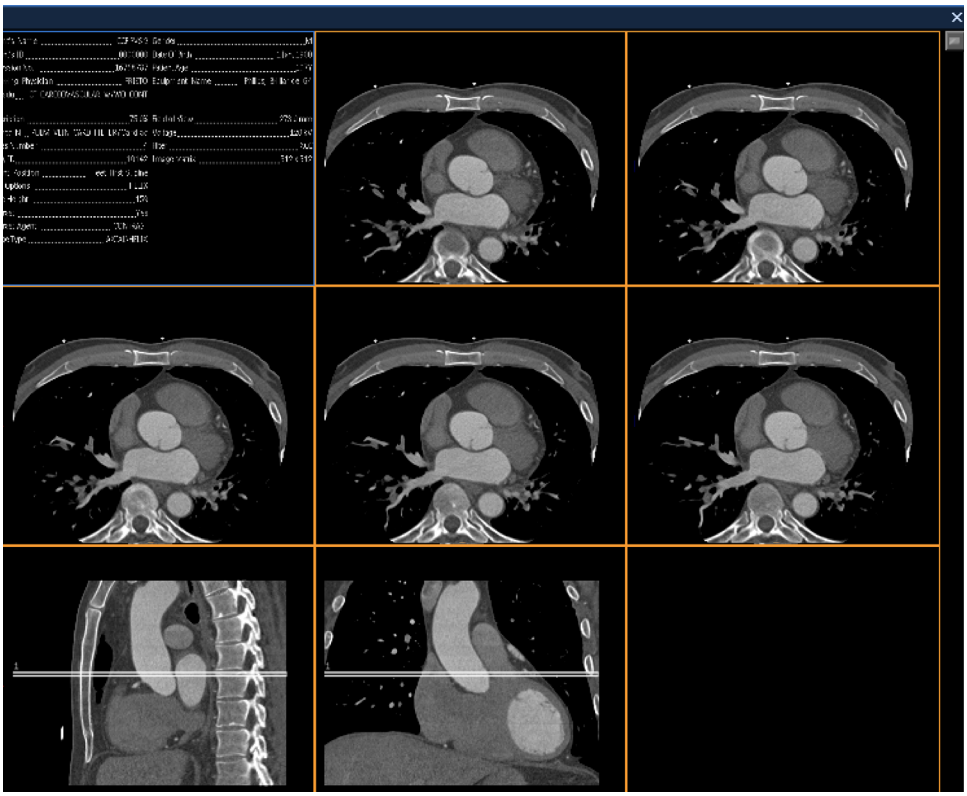
3000067186 91_A/881 * 2021-06-30

Philips

When you exit the CT viewer, a temporary series is deleted.

Preview 2D Batch

To see a preview of the batch you have created, click the Preview icon. The example preview shown below consists of 5 images in this small batch, plus one Parameters frame and two Reference images.



At the bottom of the Preview window is a tool bar allowing you to Add the batch to the Series tree and to initiate save, film, and report functions. When you click the Save, Film, Film\Save, and Report buttons, you will enter the same workflow that is described for these functions in the Common Tools section. See section “CT Common Tools” on page 141.

Create Slab Batch

The Slab batch function allows you to create a series of slab images for saving, reporting, cine and filming purposes.

In addition to the usual functions provided by batch, the Slab batch mode has particular features:

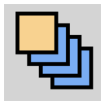
- You can specify the slab thickness and the increment between images.
- You can use Batch Preset protocols for automated batch definition.
- You can quickly “Select All” images for the batch.
- You can use Batch Preset protocols for automated batch definition.

3000067186691_A/881 * 2021-06-30

Define Slab Image Batch

The batch is created using the active viewport. For a coronal batch (for example), click the coronal viewport, then follow this procedure.

1. Scroll to the image you want as the first image of the batch.
2. Click **Mark First**.



3. After the First image has been designated, scroll to the image you want to be the last image of the batch.
4. Click **Mark Last**.

You can also use **Click All** to select all images. The full volume batch is created, using the thickness shown in the Batch tab.

Slab Batch Parameters

After defining the range of the batch, you may set additional parameters:

Thick

You can change the thickness of the slabs in the batch.

Reference images are linked, therefore they are all active when selected.

You have four options for changing the slab thickness:

- Click the viewport control for SW (slice width).
- Type a value in the thickness box.
- Click the up or down arrows in the box.
- Hold down the **<shift>** key, place the cursor on the image, and using the left mouse button drag up and down.

The default slice increment depends on the slice thickness and number of images chosen.

Increment

You can change the Increment. Changing the increment changes the number of images, but not the range of the batch.

No. Images

You can change the total number of images. Changing the number of images changes the increment, but not the range of the batch.

When saving to a large batch, there may be a memory issue (too many images and not enough memory). When this occurs, a message appears in red notifying you to reduce the number of images below a specific number to enable all batch parameters. The application automatically disables the features listed below:

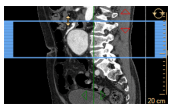
- Add Parameter Frame
- Add Reference Images
- Add Mini Image
- Tile View
- Preview

In order to use any of these features, you must reduce the number of images in the batch.

NOTICE

Since the **Add Mini Image** feature uses a lot of memory, even after reducing the number of images, a message may appear in the message area indicating that the Mini Image option was removed since the batch size exceeds available memory. If this is the case, reduce the number of images.

Slab Batch Indicator



You can change the number of images in the batch by changing the batch range. Observing the blue batch indication on the reference images, grab and move the upper and lower horizontal lines. Change the MPR plane of the batch by rotating the batch indication on the reference images. Select a different range for batch creation by moving the batch indication on the reference images.

Add Additional Images to Slab Batch

You can add other images to the batch by clicking one or more of the add tools.

These tools may be disabled if there is insufficient memory available. If one of the tools is not available, reduce the number of images in the batch.

Add Parameter Frame to Slab Batch

Click the icon to add an image of the series (study) parameters to the Batch.

3000067186691_A/881 * 2021-06-30

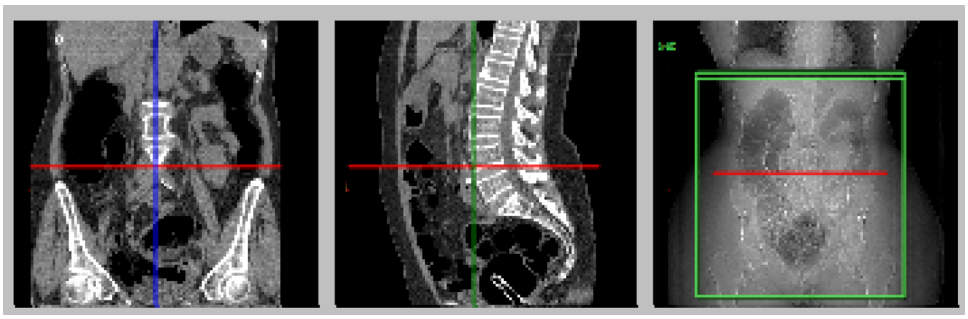
Philips

Add Reference Images to Slab Batch

Click the icon to add one or both reference images to the Batch. Select an image or select None.

Add Mini Image to Slab Batch

The Mini Image function places a small image of a reference viewport in the lower-right corner of the main image. Click the mini image icon. The Mini Image selection window opens, showing the available reference images.



Select one of the images. The image will be placed in the batch, and will appear in the viewport.

WARNING

Do not attempt to measure mini images; inaccurate results will occur.

Only one can be selected. If you do not select one, the dialog is closed without adding a mini image.

NOTICE

When using this function, a message may appear in the message area indicating that the **Mini Image** option was removed since the batch size exceeds available memory. If this is the case, reduce the number of images in the batch.

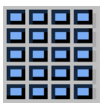
Include All Viewports in Slab Batch

Check the "Whole screen" box if you want all the viewports currently in the display to be included in the batch.

3000067186 91_A/881 * 2021-06-30

Philips

Slab Batch Tile View



Click this icon to view all images in the batch to be displayed in a static tile-sized display format. You can scroll the entire batch.

Preview Slab Batch

To see a preview of the batch you have created, click the **Preview** icon. The Preview function is the same in all viewing modes. For more information, see section “Preview 2D Batch” on page 216.

Clear Slab Batch

Use this icon to clear the current defined batch series. You can start defining a new batch if desired.

Create Volume Batch

The Volume batch function allows you to create a sequence of volume rendered images for saving, reporting, cine and filming purposes. (This type of batch is often used in Analysis applications.)

NOTICE

Some of the features of the batch tab may change, depending on the current active image (the image with the blue frame).

If the active image is a reference image, batch functions are the same as in the Slab viewer, except with no mini or reference images.

In addition to the usual functions provided by batch, the Volume batch mode has particular features:

- You can use Batch Preset protocols for automated batch definition.
- With a single click of the Quick Batch icon, you can create a rotating batch of the entire volume.
- You can manually create a batch for movies by preparing and marking each individual frame of your proposed movie. Between frames you may change viewing parameters as desired. When you are finished, the custom batch can be saved as a movie.

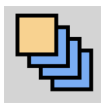
3000067186691_A/881 * 2021-06-30

Philips

- The Time Mode batch allows you to create a batch of images, viewed over time (through the heart cycle) from a single location. (Time Mode batch is available in Comprehensive Cardiac Analysis, and in the Cardiac Viewer, Slab Viewer mode.)

Prepare Volume Batch

1. Scroll or rotate the volume image to the view you want to be the first image of the batch.
2. Click **Mark First**.

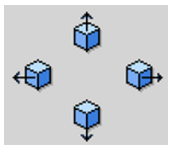


3. After the First image has been designated, scroll or rotate the volume image to the view you want to be the last image of the batch.
4. Click **Mark Last**.

You can click **Mark Last** more than once in the Volume view mode in order to create a movie.

Use the Clear icon to clear the currently defined batch.

Quick Volume Batch



This function allows you to quickly create a batch of the rotating volume image. The main parameter choices are:

- Select the direction of rotation.
- Select either the number of images between first and last or the number of increments between first and last.

Click one of the 4 icons that corresponds to the direction you want the volume to turn.

- Click the left-pointing arrow for a batch that rotates to the left.
- Click the up-pointing arrow for a batch that rotates upwards.
- Click the right-pointing arrow for a batch that rotates to the right.
- Click the down-pointing arrow for a batch that rotates downwards.

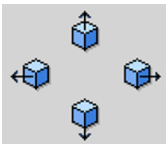
Number of Images in Quick Volume Batch

You can design a batch to extend around a given rotational arc (Degrees), consisting of a specified number of images (No. Images).

3000067186691_A/881 * 2021-06-30

Philips

Incremental Quick Volume Batch



You can design a batch to extend around a given rotational arc, with each image a uniform rotational increment (deg.) from the previous image. To access the Increment mode, click on the **down arrow** and select **Increment**. Use the up and down arrows of the combo boxes to select the number of rotational degrees, the number of images, and the rotational increment (or, type in the desired values.)

Add Parameters to Quick Volume Batch

Select this to include a parameter frame at the beginning of the quick batch, or the end, depending on what is set in Preferences.

Include All Viewports in Quick Volume Batch

Check the "Whole screen" box if you want all the displayed viewports included in the batch.

Preview Volume Batch

To see a preview of the batch you have created, click the **Preview** icon. The Preview function is the same in all viewing modes. For more information, see section “Preview 2D Batch” on page 216.

Create Endo Batch

The batch function in the Endo viewing mode allows you to batch a sequence of virtual Endoscopic images. An Endo batch consists of volume rendered images created along a path (a “curve”) within a vessel or an air or contrast filled structure.

In addition to the usual functions provided by batch, the Endo batch mode provides these features:

- Create a batch from a saved curve.
- Create a curve from navigation.

The Batch function allows you to create a series of endo images for saving, reporting, and filming purposes. It also allows you to create a flythrough path.

NOTICE

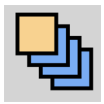
Some of the features of the Endo batch tab change, depending on the active image (with the blue frame). With active reference images, batch functions are like those in the Slab viewer, except with fewer options. See Slab Viewer in the CT Review section.

3000067186691_A/881 * 2021-06-30

Philips

Define Endo Batch with No Active Curves

1. Make the Endo viewport active
2. Click **Mark First**. The current image is the start of the batch.



3. Navigate to the desired end of the batch using the scroll function.
4. Click **Mark Last**. A virtual curve is created.

5. Specify the number of images. (You can change the number only after the batch is created.)

Increment

The user does not have the ability to change the increment. The increment is controlled by the number of images in the batch.

No. of Images

You can change the total number of images. Changing the number of images will change the increment, but not the range of the batch.

Save as Movie

A movie of the navigation can be saved to the Directory, using the **Save Results As...** feature. See section “CT Common Tools” on page 141.

Clear

Use the Clear icon to clear the currently defined batch.

Define Endo Batch on Existing Curve

In this state of the Batch function in Endo viewer: The All curve icon is enabled. In the Curve mode, Cine scrolls all viewports in unison along the path.

1. Access the Curve tab.
2. Select a curve from the curve list.
3. Access the Batch tab.
4. Click the **All curve** icon. A batch is created from the entire path of the selected curve. The Mark first and Mark last icons become disabled.

3000067186691_A/881 * 2021-06-30

Philips

5. Change the number of images, if desired. The increment adjusts, but the length of the curve stays the same.

Preview Endo Batch

To see a preview of the batch you have created, click the **Preview** icon. The Preview function is the same in all viewing modes. For more information, see section “Preview 2D Batch” on page 216.

Clear Endo Batch

Use this icon to clear the current defined batch series. You can start defining a new batch if desired.

Use Batch Presets

Batch presets can be created for some viewing modes, applications, and image types, for example Volume and Slab viewing, cMPR images, cross sectional images, and paddle wheel.

There is a separate preset list for each application and each image type. Batch presets do not transfer between applications.

The Batch Preset function allows you to create and save batch protocols that you use commonly, for faster post processing (runoff, for example).

You are able to save the orientation, thickness, increment, zoom, position (middle of batch as % of total volume), number of images, rendering, windowing, added reference or mini images, All or Whole screen options.

Create Batch Preset

1. Manipulate the image parameters as needed for the desired results.
2. Mark the First and Last batch locations.
3. Cine through batch to make sure it is what you want.
4. Click the drop down arrow next to the Preset name and select **Save Preset**. The Save Batch Preset dialog opens.
5. Name the batch and click **Save**.

View Batch Preset Parameters

Hover mouse over Preset name to see the preset’s parameters.

3000067186 91_A/881 * 2021-06-30

Activate Batch Preset

1. Activate the image on which the batch should be created.
2. Click the drop down arrow next to the Preset name. The list of saved presets is displayed.
3. Click the desired preset. The batch is created.
4. Use the Preview Batch icon to view the batch.

Delete Batch Preset

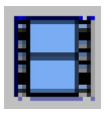
1. Click the drop down arrow next to the Preset name. The list of saved presets displays.
2. Select the Preset you want to delete.
3. Click the **Delete Preset** command.

Share Batch Presets

If you are a Clinical Administrator and are creating a new batch protocol, the Share with all users function is available. (It is grayed out for non-administrative users.) A shared batch preset becomes available to all existing and new users of the originating IntelliSpace Portal system. (Non-administrative users cannot delete, rename, overwrite, or modify shared presets).

1. Click **Share with all users**. The following message displays: “This action will affect all users. Do you want to continue?”
2. Click **Yes**. The Save Batch Preset dialog opens.
3. Type in a Preset name.
4. Leave Share with all users checked.
5. Click **Save.**

View Batch in Cine (Automatic Scrolling)



The Cine function, available in all viewing modes, allows you to view (“play”) the batch images in a cine (automatic scrolling) mode. To Cine the current batch:

Click the **Show Cine** dialog icon in the Common Tools menu (see section “CT Common Tools” on page 141). The semitransparent dialog box appears on the viewport with the Batch option already selected.

You can now view the batch in cine mode.

Click the Pause button before closing the Cine dialog box, or the images will continue to cine.

Create Batch Movie

You can create a batch of a volume and save it as a movie file. To create a batch for making a movie, manipulate your image as desired, for example by changing the zoom, pan, and windowing parameters, and changing protocol, rotation, etc. After you have created the batch, follow the procedure below.

NOTICE

To save movies, your IntelliSpace Portal client computer must be equipped with Windows Media Player.

When you save a Batch Movie, any movement of the volume, or change in protocol or windowing, or change of bounding tools, that you made between stops is interpolated.

1. Select **Save Batch As...** in the Common Tools. See section “CT Common Tools” on page 141.
2. Enter a name for the movie in Description, if desired.
3. Select:
   - **Movie** from the Format drop-down to save as a WMF file.
   - **AVI** from the Format drop-down to save as an AVI file.
4. Select the movie’s Quality using the slider (higher quality requires more storage space).

Movies are saved in the Files tab in the Series List. You can export the movie using the right click menu in the Files list.

Tissue Management Functions

The Tissue Management function is available in the Slab, Volume and Endo viewing modes.

The Tissue Management function allows you to control the display of the volume image.

The Management function lists the tissue definitions that have been created for the current study. The list includes tissues defined in the current work session as well as those defined during previous work sessions, and from other applications (if they are loaded with the study in the form of Results).

Many tissues manipulations can be associated with the current study, including:

- Bone removal.
- Sculpting.

3000067186691_A/881 * 2021-06-30

Philips

- Segmentation of various structures, such as vessels, liver and kidney, airways, as well as volume tracing.

Tissue Management List

The Tissue Management tab lists all current and all saved tissues of the study.

Volume and Couch

Volume is the original volume comprised of all the slices. Couch allows you to show or hide the couch.

WARNING

Verify couch and head holders were correctly removed.

Name

Each tissue is identified by its name.

Show or Hide Tissues

Each active tissue can influence the volume in two ways: Show and Hide.

Show

When the Show cell is checked, the part of the volume defined by the tissue is visible on the view port.

Hide

When the Hide cell is checked, the volume defined by the tissue is subtracted from the volume shown in the view port. (If you hide a tissue, its rendering parameters are grayed out.)

NOTICE

The tissue selection and the Show/Hide state in the list affects each of the image viewports on the screen separately. This means that you first need to select the viewport and only then check/un-check the tissues in the list.

Tissue Selection

Clicking the checkbox of a tissue makes it visible in the image display. Any number of tissues may be checked to make them visible.

3000067186 91_A/881 * 2021-06-30

Philips

If you want to manipulate a tissue (change tissue color, change volume rendering presets), make the tissue “active” by clicking its name in the list.

The active tissue name is highlighted blue in the tissue list. Only one tissue can be made active at a time.

By default, Volume is the selected item when the dialog is opened.

You can change the volume rendered protocol of the selected tissue by using the Change Preset selection in the Tissue Management box, or the viewport controls.

NOTICE

The size and shape of anatomies can change when tweaking a protocol. To prevent wrong interpretation, follow the Change Preset procedure for the application you are working with.

Select Tissue Color

Click this icon to select a color for the tissue. A dialog box opens where you can select a color from a matrix of pre-defined colors. If desired, you can define new colors by selecting “Define Custom Colors” and setting HSL or RGB values.

Change Preset

Click this icon to display the Volume Rendering Presets window, from which you can select a different rendering protocol than what the system chose by default. The Change Preset icon is grayed out in Slab viewing mode, where Volume Rendering is not available.

Fill Tissue

The Fill Tissue function adds to the injected segmented tissue, filling in any holes within the segmentation volume.

Expand Tissue

The Expand Tissue function allows you to increase the edges of the segmented tissue. Each click expands the edge by a one-voxel increment.

3000067186 91_A/881 * 2021-06-30

Philips

Erode Tissue

The Erode Tissue function allows you to decrease the edges of the segmented tissue. Each click reduces the edge by a one-voxel increment.

Reset

Click this icon to return current selected tissues to their previous state.

Save Tissue Results

Use the "Save Results As..." function in the Common tools to save tissues. Note that when you save tissues, you are saving all changes and all tissues shown on the display. See section “CT Common Tools” on page 141.

The CT Viewer application allow saving tissue results in RT Structure format for export. Select the RT structure option from the save dropdown menu.

The Viewing Application preference, **Save RT Structure Sets as dense contours** in System Preferences also provides an option to use the saved RT structure sets as dense contours.

Layout Manager

NOTICE

The Layout Manager is only relevant for CT applications. In all CT applications the last used layout is used next time the application is opened by the same user.

The Layout Manager dialog allows you to manage layouts (viewport display arrangements).

3000067186691_A/881 * 2021-06-30

Philips

Different viewers and applications have different layouts. With the Layout Manager, you can change the current layout as desired and save it as a new layout. You can also export and import layouts to and from other IntelliSpace Portal systems.

Default Layout

The system uses this icon to identify which layout is the default.

Add Layout

Click this icon to access the list of factory layouts. This function allows you to return a factory layout to the list if it has been previously or accidently deleted from the list.

Delete Layout

Select the layout from the list that you want to delete and click this icon. Deleted user-defined layouts are removed permanently.

Export Layout

Click the icon to open the Export or Import browser window. Using the browser, you can export a layout to a CD disk, USB device, or a network location.

Import Layout

Click the icon to open the Export or Import browser window. Using the browser, you can import a layout to a CD disk, USB device, or a network location.

Rename Layout

Use this function to rename the selected layout. (A factory layout cannot be renamed.)

3000067186691_A/881 * 2021-06-30

Philips

Create Custom Layout

Arrange the images on the screen.

Within the various applications you can swap between the viewports by dragging with the right mouse. Also, you can select a different type of image to display using a right click menu. (For example: in the Virtual Colonoscopy application you can right click on a certain viewport and select which type of image to display in this viewport.)

After you have arranged the images on the screen in the viewports as desired, go to the layouts drop down list and select the **Save layout** option. The layout is added to the Layouts drop down list and to the Layout Manager.

CT Pre-processing (Option)

Pre-processing is an available software option that allows time-intensive image processing to be done in the background while you use the IntelliSpace Portal to perform other viewing and analysis activities. The following CT applications support pre-processing:

- AVA - Skull Removal
- AVA - Bone Removal
- VC - Colon Segmentation
- CCA - Heart Segmentation
- Liver - Segmentation

Manual Pre-processing

Manual Pre-processing is a function you activate at the same time you launch a series from the Patient Directory.

1. Access the Series list (the Series tab) in the Patient Directory.
2. Right click on the series to be Processed. The selection list appears. Manual pre-processing is performed on one series. If you try to select multiple series, the options are greyed out.
3. Select the appropriate Pre-processing function.
4. Processing begins in the background. The Pre-processing tab in Queue Manger allows you to view progress, change the order, retry, or delete the cases.

NOTICE

The Directory also shows the number of studies being processed for the current user and the total number of studies being processed for all users.

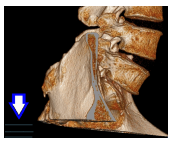
3000067186 91_A/881 * 2021-06-30

Philips

Automatic Pre-processing

Automatic Pre-processing is a function you set up in Preferences. It performs the same time-intensive processing as Manual, but earlier: Pre-processing is performed when the patient dataset is being copied to the IntelliSpace Portal from the network. Pre-processing is then not needed during launch.

Image Rendering Mode



To select the desired rendering mode, click on the rendering mode name in the rendering Viewport Control (located in the bottom left corner of the image) in the viewport. A pop-up menu allows you to select a new mode.

NOTICE

Slab and volume images may have different rendering modes available. For example, the "Average" mode is not available on volume images.

Average Image Rendering

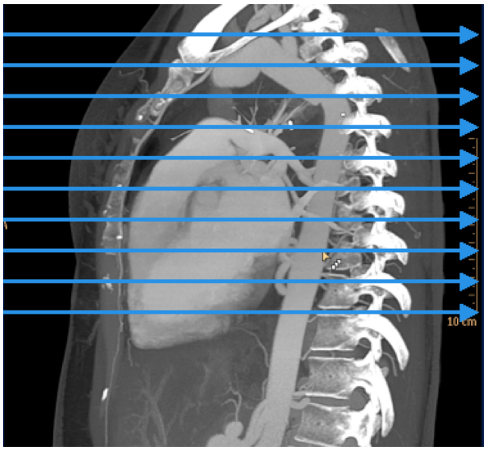
Average is the default rendering mode for MPR images in most of the applications. Average renders the volume image according to the average value along the path (ray) through the patient (perpendicular to the screen).

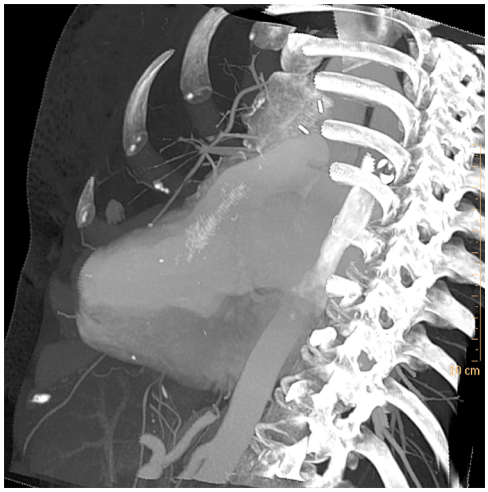
Maximum Intensity Projection (MIP) Image Rendering

MIP displays the highest intensity (maximum intensity Hounsfield unit, in the case of CT) voxel along a particular projection through the volume dataset (perpendicular to the screen). This creates a 2D image composed of the brightest voxels.

3000067186 91_A/881 * 2021-06-30

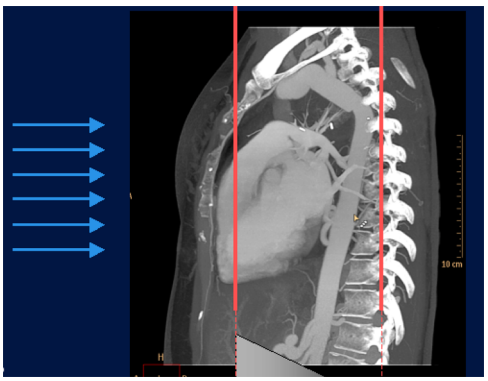
Philips

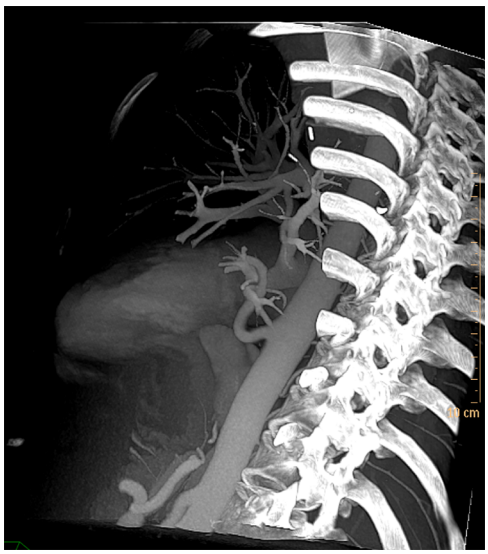




Volume Intensity Projection (VIP) Image Rendering

Volume Intensity Projection, like MIP, displays the highest intensity voxel along a projection through the volume dataset. Unlike MIP, the VIP mode assigns the highest intensity, or greatest brightness value, to voxels closest to the eye-point, while those farther away are faded according to their distance from the eye-point.





The Center value controls the location of the ramp over the volume. The Width value controls the rapidity of the fade.

You can change the Center, Width, and other values by clicking on their display lines.

Use the slider and up/down arrows to change the setting.

3000067186691_A/881 * 2021-06-30

Minimum Intensity Projection (MinIP) Image Rendering

The MinIP image displays a 2D image of the lowest intensity (minimum intensity Hounsfield unit, in the case of CT) voxel along a particular projection through the volume dataset (perpendicular to the screen). One of the possible uses of MinIP images is to demonstrate respiratory airways.

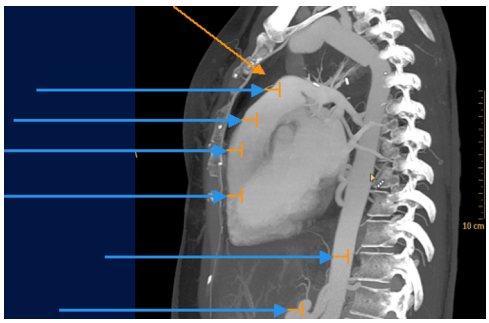
Default threshold is -1000 or -1024 HU, depending on the scan. (This setting ignores air). The range is -1024 to 3071. You can change the Threshold value by clicking on its display line.

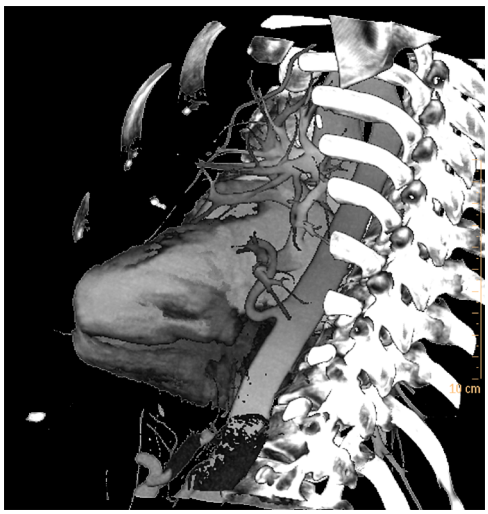
Type in the desired value or click the up or down arrow buttons.

Surface MIP Image Rendering

Surface MIP rendering allows you to generate volume images by detecting tissue surfaces based on the Distance (display depth) and Threshold (Hounsfield Units).

When the projection ray hits a voxel having an HU value equal to or larger than the threshold, a MIP calculation is started for the distance (mm) specified.





You can change the Distance value by clicking on its display line.

Type in the desired value or click the up or down arrow buttons.

You can change the Threshold value using the same method as Distance.

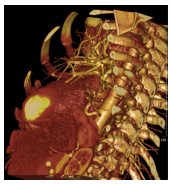
Volume Image and Photorealistic Rendering

WARNING

The volume image displays the anatomy according to the defined protocol. Do not use the volume image as the SOLE basis for a diagnosis.

3000067186691_A/881 * 2021-06-30

Philips



Volume rendering assigns opacity and color to every voxel according to rendering protocols for vasculature, soft tissue, and bone. Selected structures can be rendered opaque, translucent, or invisible, allowing visualization through and beyond contiguous structures.

You can change the current volume rendering protocol by clicking on the current protocol name in the viewport and then either selecting a protocol from the predefined protocols list or using the Edit preset option to edit the protocol.

High Quality

This function adjusts the rendering parameters to display a sharper image to enhance details. Activating High Quality can slow processing. It is best to complete all image processing before activating it.

Opacity

The opacity range is from 0 to 100, where 0 is completely transparent, and 100 is opaque.

The default values of opacity and range are determined according to the default protocol.

You can change the Opacity value by clicking on its Viewport Control display line and dragging the slider to the desired opacity value.

Type in the desired value or click the up or down arrow buttons. You can also change the opacity by dragging the middle mouse.

Angle Display

This display shows the angle in space (alpha, beta, gamma) of the rendered object. Click the display to enter the viewport control to modify the coordinate parameters.

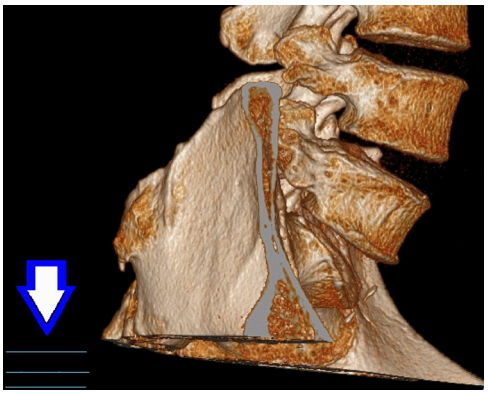
Presets

An extensive catalog of Volume Rendering protocol presets is available.

NOTICE

The default protocol for the current study is the Factory default, unless a user has changed it. You can create your own Volume Rendering Presets.

To access the catalog: Click the Viewport Control protocol link in the lower left of the viewport.



You can also click the **Change Preset** icon in the Tissue Management tab.

The VR Protocol window opens at the left side of the viewports. Protocols are separated into groups. This window displays color icons representing the available rendering protocol presets in the current VR Protocol group.

The active protocol has a blue frame around its icon. Click on a different icon to activate another preset. The volume image updates accordingly.

You may select a different protocol preset group.

Click the down-arrow to view a list of the available protocol groups. For example, if the BONE group is active, the presets in that group will be shown as icons in the VR Protocol window.

Edit Presets

Use this function to create a new Volume Rendering preset. This icon opens the Protocol Editor. See section “Protocol Editor” on page 237.

Save Preset/Save Preset As...

Select the desired Save action from the drop-down list. Use **Save Preset** to saves the active preset by its current name (only works with user defined presets). A message asks if you want to overwrite the existing preset. Use **Save Preset As ...** to save the active preset by a new name. A dialog opens for typing the new name. See also section “Share Presets” on page 237.

3000067186691_A/881 * 2021-06-30

Philips

Save Presets as Default

The active protocol is saved as the default for the current protocol group.

Share Presets

If you are a Clinical Administrator and are saving a new Volume Rendering preset, the **Share with all users** function is available. When a Clinical Administrator shares a preset, it becomes available to all existing and new users of the originating IntelliSpace Portal system. (Non-administrative users cannot delete, rename, overwrite, or modify shared presets). See also section “Save Preset/Save Preset As... ” on page 236.

NOTICE

Saving a protocol preset saves it to the current protocol group. You cannot delete, rename, overwrite, or modify a factory preset.

Remove Presets

Deletes the active preset. A message asks you to confirm the deletion.

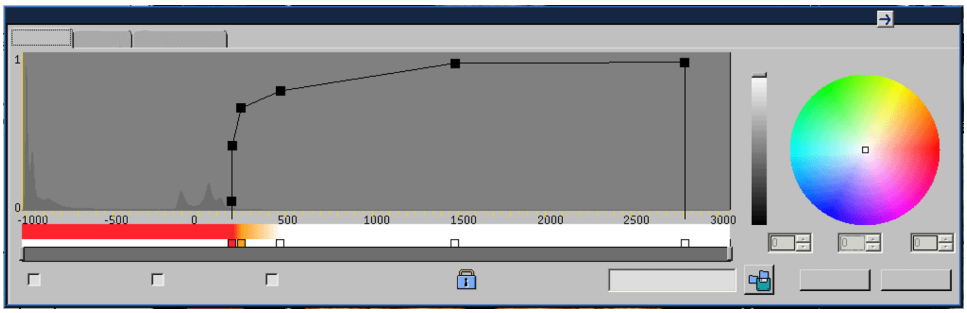
Rename Presets

Rename the active preset. A dialog opens for typing the new name.

Protocol Editor

After you click the Edit Preset icon, the Protocol Editor opens at the bottom of the viewport with the current protocol. The protocol’s name appears in the title of the dialog.

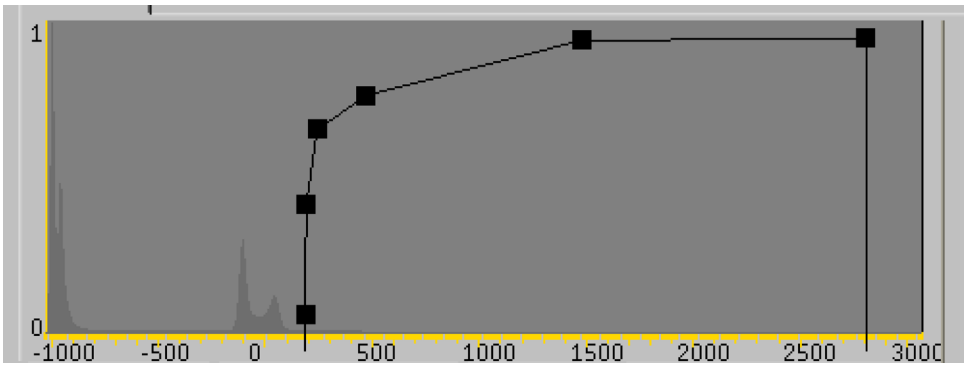
3000067186691_A/881 * 2021-06-30



The dialog functions interactively; any change you make in the editor is immediately reflected in displayed image.

Opacity Curve

Each preset has one continuous opacity curve along the whole CT Values histogram. (The opacity curve is independent and is not connected to the color scale.)



The opacity curve is a line which represents the opacity of the CT values along it. The highest point (vertically) is 100% opaque and the lowest point is 100% transparent.

Opacity Curve Control Points

Control points are small squares that appear along the opacity curve that are used to manipulate the curve. The right-most and the left-most control points are located at the beginning and at the end of the histogram (-1000 and 3095 HU) and can be moved vertically or horizontally (you must zoom the scale to see units above 3000).

Move Opacity Curve Control Points

When you move the mouse cursor over a control point, the cursor becomes a pointing arrow.

- You can move a control point by clicking it and dragging it with the left mouse button. (A selected control point has a red border.)
- Moving a control point can be done only between the two adjacent control points.
- Right clicking anywhere on the curve adds a new control point at the location where you clicked.

3000067186691_A/881 * 2021-06-30

Philips

- While dragging a control point, a tool-tip appears and shows its current CT value and opacity.

Delete Opacity Curve Control Points

Click the control point you wish to delete, to make it active. (The active point has a red frame.) Press the DELETE key to delete the selected control point. The line between the two adjacent control points becomes a joined straight line.

Move Opacity Curve

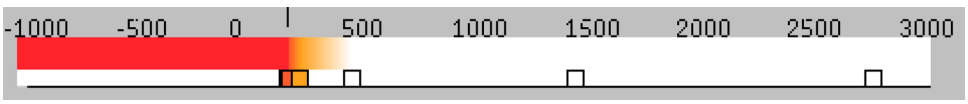
To move the whole opacity curve, drag any part that is not a control point with the left mouse button.

The opacity curve can be moved and extended outside the border of the preset editor along X-axis only and not more than 2000 CT values.

Moving the curve upward/ downward changes the opacity level of the whole curve.

Color Scale

The color scale appears below the histogram. The color points that appear below the color scale correspond to the control points in the histogram.



The color scale appears below the histogram. The color points that appear below the color scale correspond to the control points in the histogram.

Right clicking on the color scale adds color points to the scale.

You can move a color point by clicking it and dragging it with the mouse. (The mouse cursor changes into a pointing arrow. A selected color point has a red border.)

Dragging a color point changes the color gradation. (You cannot drag one color point across another color point.) While dragging a color point, a tool-tip appears, showing the current CT value of that point.

You can move the entire color scale by dragging it with the left mouse button.

If you click the Color to BW button in Common Tools, the color scale becomes grayscale, but the color points keep their colors. See section “CT Common Tools” on page 141.

Delete Color

Click the color control point you wish to delete, to make it active. Press the DELETE key to delete the selected color point. The line between the two adjacent control points becomes joined.

3000067186691_A/881 * 2021-06-30

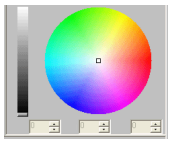
Lock Opacity and Color Scale

Use the lock mode to lock together the opacity curve and the color scale. When locked together, the color scale moves with control points of the curve. Every control point on the opacity curve is given the color that is below it on the color scale.

When you drag the whole curve, the color scale is dragged with it, and vice versa.

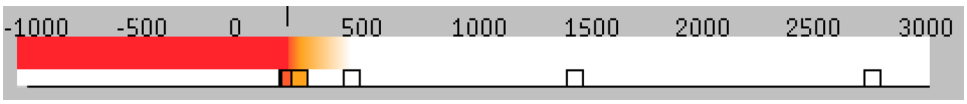
When you drag a color point on the color scale, the control point on the curve moves with it, and vice versa.

Use Color Palette

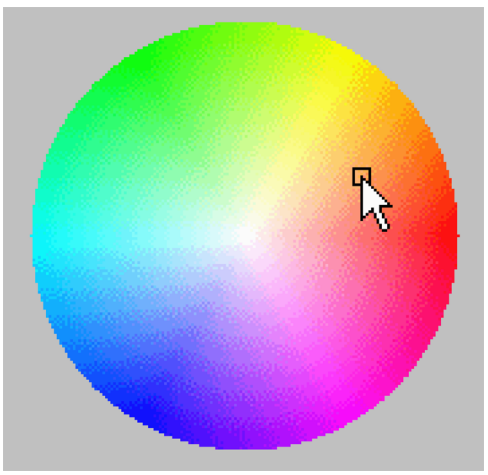


In Volume Rendering, use the color palette to change the volume image, the color bar, the color button control point and the color scale to the selected color. The color palette has a round Hue/Saturation color wheel and a gradient (Value) slider bar. A control point (the small box) appears on the wheel at the color of the selected color point. To change the color setting:

Click on one of the color control points (the colored rectangles at the bottom of the Protocol Editor dialog).



Click on the desired color on the palette.



In Photorealistic Rendering, the hue/saturation color wheel is divided into two color palettes; one palette is the Scattering color wheel and the other is the Absorption Color wheel.

NOTICE

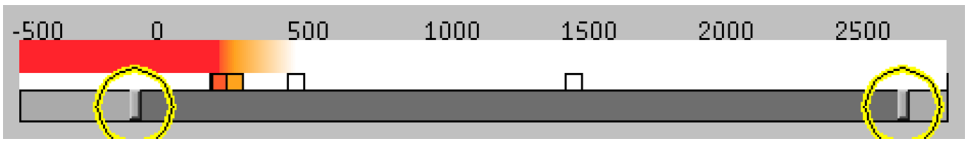
You can also use the combo boxes to set the Value, Hue, and Saturation.

300006718691_A/881 * 2021-06-30

Philips

Zoom and Pan Slide Bar

The zoom-pan slide bar is located under the histogram.



At each end of the slide bar is a vertical control slider.

To zoom in the histogram (only the width), move the slider towards the center.

To pan the histogram after zooming, point the cursor anywhere on the bar between the two vertical sliders and drag it.

Log Scale

Shows the CT Values Histogram in logarithmic mode (or the relevant values according to the relevant imaging modality).

Spline

Turns the curve into spline line.

Color Scale

Show/hide the color curve.

Protocol Name

Displays the current protocol's name. You can create a new protocol by typing in a new name.

OK Button

Closes the Protocol Editor, keeping the changes made for the volume image, but the protocol is not saved.

Cancel Button

Closes the Protocol Editor without saving any changes. The volume image is displayed according to the selected protocol.

Lighting Editor

The Lighting feature allows you to apply an illumination effect and to control the lighting parameters of the volume image. Click the Lighting tab to open the lighting editor, shown below.

3000067186691_A/881 * 2021-06-30

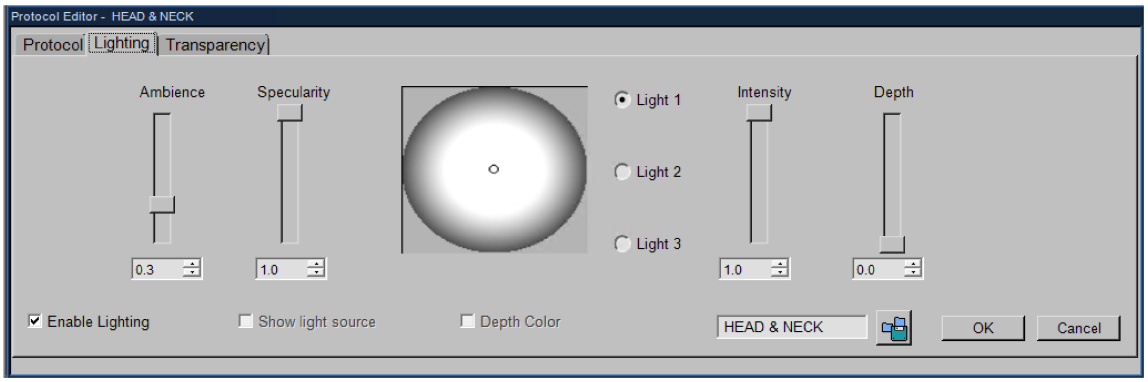


Fig. 1: Lighting for Volume Rendering

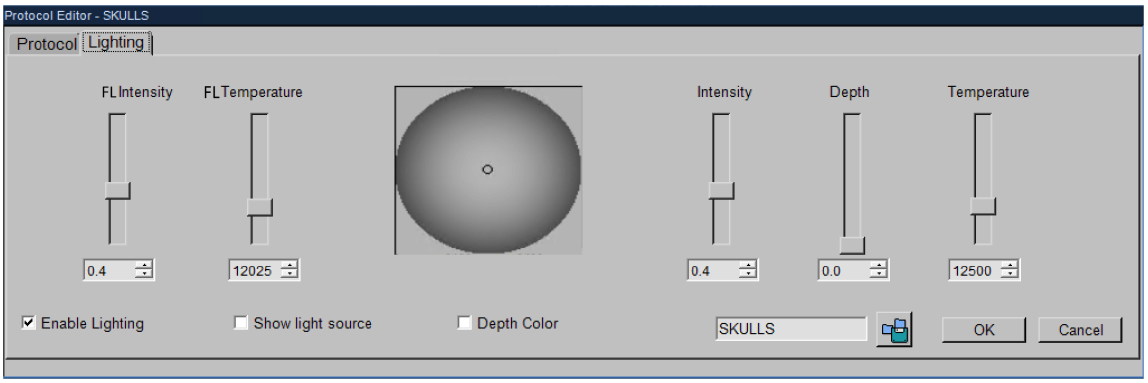


Fig. 2: Lighting for Photorealistic Rendering

All changes are displayed interactively on the volume image. Drag the parameter slide bars and observe the volume image to locate the light source.

Enable Lighting

Click the check box to activate (and deactivate) the lighting effect on the volume image according to the parameters currently set in the lighting editor.

Volume Rendering- Light 1, Light 2, Light 3

The **Intensity** parameter can be defined for three light sources. Click the button(s) to turn each light on or off.

Photorealistic Rendering Light

In Photorealisitic Rendering, there are two types of light sources with **Intensity** parameters that can be defined:

- Point source light
- Front light

Show Light Source

In Photorealistic Rendering, enabling the **Show light source** demonstrates the point source light.

This is disabled for Volume Rendering.

Depth Color

In Photorealistic Rendering, the Depth color emphasizes depth by using other colors (blue).

This is disabled for Volume Rendering.

Intensity

The range of intensity is 0.0 to 1.0. Use the slider to adjust.

Temperature

In Photorealistic Rendering, the range of Temperature is 1000 K to 40000 K.

FL_Intensity

In Photorealistic Rendering, the **FL_Intensity** is the front light intensity, with a range of 0.0 to 1.0.

Use the slider to adjust.

This is disabled for Volume Rendering.

FL_Temperature

In Photorealistic Rendering, the **FL_Temperature** is the front light temperature, with a range of 1000 K to 40000 K.

Use the slider to adjust.

This is disabled for Volume Rendering.

Depth

The **Depth** range of intensity is 0.0 to 1.0.

Use the slider to adjust.

Specularity

In Volume Rendering, specularity is “shininess.”

The range of specularity is 0.0 to 1.0 (maximum specularity is 1.0). Use the slider to adjust.

3000067186 91_A/881 * 2021-06-30

Ambience

In Volume Rendering, this slide bar controls the intensity of the ambient light source.

The range of ambient light is (0.0 to 1.0).

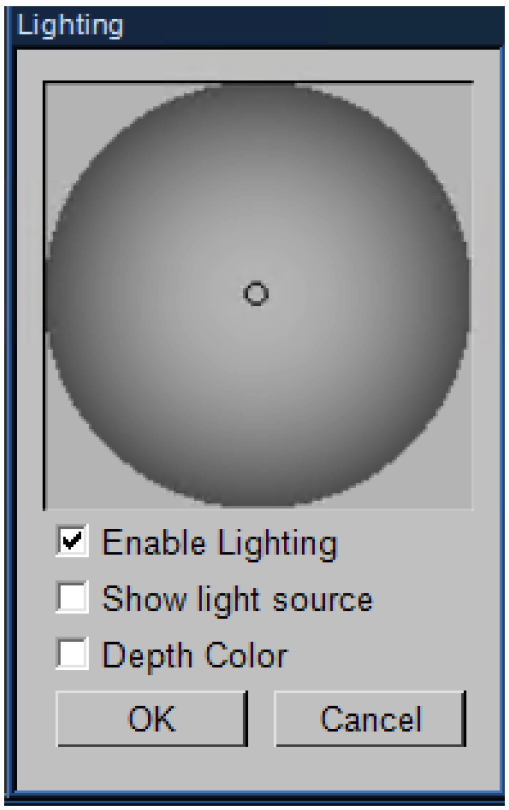
Ambience is turned off at 0.

Ambience at maximum intensity is 1.

Mini Light Source Widget

The **Mini light source** widget controls the light without opening the entire preset editor (see section “Edit Presets” on page 236). It has the following functions as in the Lighting tab (see section “Lighting Editor” on page 241):

- Enable lighting
- Show light source
- Depth color
- Controlling the light position (light wheel)



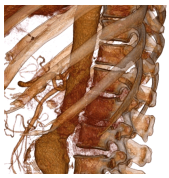
This is disabled for Volume Rendering.

300006718691_A/881 * 2021-06-30

Mouse Functions

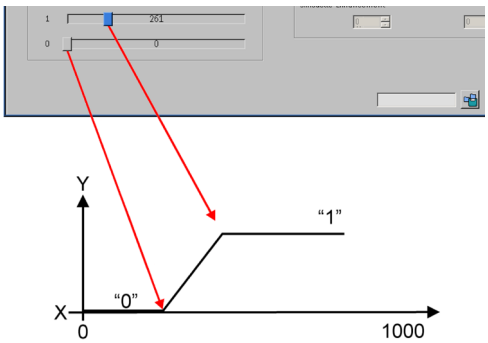
- Pressing and dragging the middle mouse button upward or downward moves the curve to the left or right, showing a different range of HU.
- Pressing the middle mouse button and dragging the mouse is the same as being in the lock mode.
- Pressing the middle mouse button and dragging the mouse right or left widens or narrows, respectively, the opacity curve.

Transparency



A transparency gives structures a "transparent-like" look, as shown in the example. The Transparency function allows you to adjust the transparency and to enhance the opacity and shading of the image.

In the diagram below, X = Gradients Magnitude, and Y = Multiplier.



To use Transparency:

1. Click the **Use Enhanced Transparency** check box.
2. Set the Boundaries Enhancement (gradient) between 0 and 1024 by sliding the bar.
3. Set the Opacity and Shading between 0 and 10 if desired.
4. Click **Save preset** (change the name if a factory protocol was used).
5. Click **OK**.

Sort

The Sorting function is available in CT Viewer - 2D mode, and in Quick Review. The Sort... function allows you to sort images using DICOM tags.

You can use factory defined sorting protocols or you can create your own by choosing Custom from Select sort. You can preview, save and delete sort options.

1. Click the **Sort** icon. The sorting dialog box appears.
2. Use the **Select sort** function to view the list of available sorting protocols (and the Custom function).

   You can also click in any of the Sort by fields and select desired parameters.
3. Set the desired Priority. The highest priority is 1; the lowest is 5.
4. Set the sort Direction. The up arrow denotes ascending sort and the down arrow denotes descending sort.
5. Click **Preview** to perform the sort without closing the dialog, allowing you to examine the results.
6. Click **OK** to accept the sort. The dialog closes and all changes that were done are saved.

Create a Custom Sort Preset

1. Select **Custom** from the Select sort drop down.
2. Select Sort by and the direction.
3. Enter the name in the blank Select sort field.
4. Click **Save sort**.

Save and Access Custom Sort Preset

You can save a preset for sorting and access it later from the Sort dialog box.

Delete a Custom Sort

NOTICE

Be careful when deleting. Factory sort presets can be deleted.

Click **Delete sort** to delete the current preset from the list. A dialog box appears, asking if you want to delete this sort.

Click **Yes**.

3000067186691_A/881 * 2021-06-30

Philips

Combine

The Combine function combines groups of original thin “slices” into fewer, thicker “slices” for viewing, filming, reporting or saving. The Advanced Combine function allows you to manipulate the basic combined sets of images by adding them to each other or subtracting them from each other.

NOTICE

Combined images should not be used as the SOLE basis for clinical diagnosis.

Perform Basic Combine

The basic Combine function combines groups of original thin “slices” into fewer, thicker “slices” for viewing, filming, reporting or saving.

NOTICE

Combined images should not be used as the SOLE basis for clinical diagnosis.

3000067186 91_A/881 * 2021-06-30

The Combine function operates on every image of the selected series. Images are combined in the order they appear in the viewport.

The "Combine every:" field allows you assign to an integer number value for the number of sequential images you want to combine.

1. Select the series that you wish to combine from the Series tab.
2. Click the drop-down arrow next to the **Combine every:** icon and select the desired Combine value. (You can also click in the field and type in the value and press <Enter>).
3. The series is combined. Images that remain at the end of the series that are fewer than the integer value are not combined.

Basic Combine Example

As an example, if the order of the images that you want to combine is 1, 2, 3, 4, 5, 6, etc., and you select “Combine every: 3,” the result is:

- The new image 1 is created from original images 1, 2, and 3.
- The new image 2 is created from original images 4, 5, and 6.
- Etc.

The word “Combine” is added to the image label (in the upper left-hand corner) for the new combined images.

Philips

Perform Advanced Combine

The Advanced Combine function allows you to manipulate the basic combined sets of images by adding them to each other or subtracting them from each other (within each combined image set). The primary use is to perform subtraction on contrasted and non-contrasted studies. You can also have the basic combined images displayed in a way that shows, in each pixel location, either the minimum (“Min”) or the maximum (“Max”) pixel value.

After you have combined a series of images, the Advanced Combine icon becomes active. Clicking the Advanced Combine icon activates the function. The Advanced Combine dialog box displays.

NOTICE

Combined images should not be used as the SOLE basis for clinical diagnosis.

Combine Every Option (5 or Fewer Images)

By default, when the Combine Images dialog box opens with "Combine every:" set at 5 or fewer, the weight fields are active, and you can enter the desired weight values.

NOTICE

Combined images should not be used as the SOLE basis for clinical diagnosis.

Combine Every Option (6 or More Images)

By default, when the Combine Images dialogue box opens with "Combine every:" set at 6 or more, the weight fields are active, and you can enter the desired weight values.

NOTICE

Combined images should not be used as the SOLE basis for clinical diagnosis.

Combine Maximum or Minimum Operation

Select one of these operations to produce a series of 2D images based on either the minimum or the maximum value of each of the original pixels that were combined to form the new images. A Minimum image is a 2D image of the lowest intensity pixels and a Maximum image is a 2D image of the brightest pixels.

3000067186691_A/881 * 2021-06-30

Philips

NOTICE

Combined images should not be used as the SOLE basis for clinical diagnosis.

Combine Sum Operation

Select this operation if you want to create a single “added” or “subtracted” image from each of the combined images in the series.

NOTICE

Combined images should not be used as the SOLE basis for clinical diagnosis.

Weighted Images

This parameter determines whether an image can be added or subtracted. A numerically positive weight causes the image to be added. A numerically negative weight causes the image to be subtracted.

NOTICE

Combined images should not be used as the SOLE basis for clinical diagnosis.

Equal Weight Images

When the dialog box is first opened (when you are working with 5 or fewer combined images) the system equally distributes the weighting among the images. All weight values are numerically positive, and their sum equals 1.00.

NOTICE

Combined images should not be used as the SOLE basis for clinical diagnosis.

Add Weighted Images

When adding images (up to 5), you can assign weight values between -100 and +100. A greater weight causes an image to have greater influence on the final, summed image. A lesser weight has lesser influence.

3000067186 91_A/881 * 2021-06-30

Philips

NOTICE

Combined images should not be used as the SOLE basis for clinical diagnosis.

Subtract Weighted Images

Subtraction is usually used to subtract 2 studies of the same patient: one study is non-contrasted and the other is contrasted. To select this subtraction assign weights of +1.00 and -1.00, respectively.

NOTICE

Combined images should not be used as the SOLE basis for clinical diagnosis.

Normalize Weights

Normalization is a feature that proportionally adjusts the weights you assign, so that the resulting images display in the normal CT scale. (Without normalization, if the sum of the weights you assign is greater or less than 1.00, the resulting CT values are biased, causing the image to be darker or lighter than normal.)

NOTICE

Normalization SHOULD BE USED when adding series (when all weights are zero or greater).

Normalization SHOULD NOT BE USED when subtracting series (when weights are both positive and negative).

If the sum of all weights equals 1.00, normalization does not influence the results.

NOTICE

Combined images should not be used as the SOLE basis for clinical diagnosis.

Advanced Combine Bias

Use the Bias parameter to add a constant CT value to the resulting images. The default Bias value is 0 (zero). The Bias drop-down list offers these values: 0, 24, 1000, and1024. You can also type a Bias value between -1000 and 3095 for CT (or between -4095 and 4095 for MR). Bias can be changed at any time, before or after you see the results.

3000067186691_A/881 * 2021-06-30

Philips

NOTICE

Combined images should not be used as the SOLE basis for clinical diagnosis.

Advanced Combine Preview

Use Preview to perform the Advanced Combine and view the results without closing the dialog. You can then adjust parameters and perform the combine as often as desired.

NOTICE

Combined images should not be used as the SOLE basis for clinical diagnosis.

Save or Cancel Advanced Combine

Use OK to close the Advanced Combine dialog and save all settings and changes you made. Use Cancel to close the Advanced Combine dialog without saving the combine results. The data reverts to its state before the dialog was opened.

NOTICE

Combined images should not be used as the SOLE basis for clinical diagnosis.

Subtract Images

The following procedure describes how to use the Advanced Combine function to subtract 2 studies of the same patient, one pre-contrast and one with contrast.

NOTICE

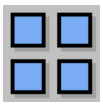
Combined images should not be used as the SOLE basis for clinical diagnosis.

When subtracting images, it is important for the images to have the same Z location to achieve accurate results. The Z location can be off by a small amount, but results will not be not optimum. In steps 8 and 9 below, you are advised of your options.

1. From the Directory, select a study without and with contrast, each having the same number of images.
2. Load the study to the CT Viewer.
3. Select the 2D Viewer.
4. Click the **All selection** icon.

3000067186 91_A/881 * 2021-06-30

Philips

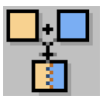


5. Make the non-contrast study “active” by clicking on it in the series tree. It is highlighted in black.
6. Click **Sort**. The Sorting box opens.

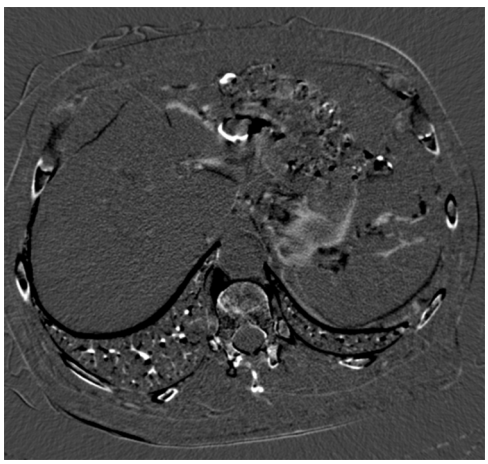
7. Click the **Select sort** icon. A drop-down list opens.
8. If your Z locations are accurate, select Mixed Series (Z pos.).

   If your Z locations are off by a small amount, select Mix Series (Image#).

   Make sure the pre-contrast series is the first image, then click OK.
9. Set **Combine every:** to the value 2. The Advanced Combine icon becomes active.
10. Click the **Advanced Combine** icon. The Combine Images dialog box showing 2 Weight fields opens.



11. To perform subtraction, change the Weights to 1.00 and -1.00.
12. Uncheck **Normalize weights**.
13. Adjust Window and Center as desired, and change Bias as desired.
14. Click the **Preview** to see the result. Below is a subtracted image of the study shown above. The bias for this image is set at 1000.



15. Click **OK** to close the dialog and save all changes that were done.

    Or:

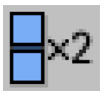
    Click **Cancel** to close the dialog without saving the combine results. The data reverts to its state before the dialog was opened.

3000067186691_A/881 * 2021-06-30

Philips

Combine Images in Different Scenes

You cannot use combined images in other scenes unless you save and load them from the Directory.



Perform the Combine function. Click **Duplicate**. This creates a temporary series (of the combined series), which can be viewed in any of the scenes.

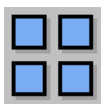
NOTICE

Combined images should not be used as the SOLE basis for clinical diagnosis.

Combine Two Series

You cannot use combined images in other scenes unless you save and load them from the Directory.

1. Perform the Combine function.
2. Click the **All selection** icon.



3. Click the **Sort** icon. The Sort dialog box opens.

4. Select **Custom.**
5. Set the first priority to Image number.
6. Click **OK**. The 2 series are combined into 1.

NOTICE

Combined images should not be used as the SOLE basis for clinical diagnosis.

Compare

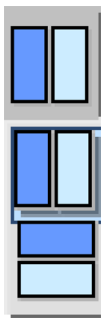
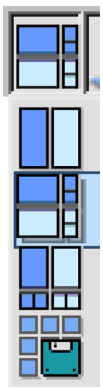
The Compare function (available in the 2D, Slab, Volume, Lung Density, AVA, and Virtual Colonoscopy viewing modes) allows you to perform a side-by-side review of selected images.

If there is only one series loaded in the viewer, a duplicated series is automatically created when Compare is selected (2D and Slab only).

If you have two series, you have a choice of vertical or horizontal layouts.

3000067186691_A/881 * 2021-06-30

Philips

2D Layout	Slab Layout
	

Both series are selected and shown side by side in Compare mode.

Three selection modes are available in the compare mode: (2D only).

- **Image mode.** Functions in the same way as a series mode.
- **Series mode.** One series is selected. Click on the series to be scrolled.
- **All mode (default).** Used to scroll together all of the compared series.

NOTICE

The series to be compared are the ones highlighted in the series tree, regardless of the selection mode. If you select a series that already has a duplicate (twin), and you click Compare, the series is not duplicated again. Instead, the twin is automatically selected for the compare mode.

Compare Linked Images

When using the Compare function in the Slab viewing mode, the Link icon allows you to link images (“lock” together) to perform the same manipulation on the image(s) of your choice, such as manipulating, filming or saving.

Compare in Spread Mode

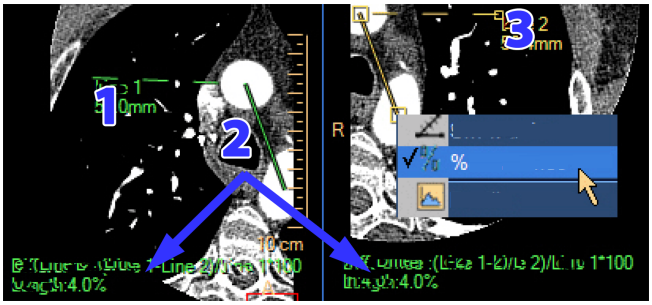
When the Compare mode is active, you can click the Spread icon to spread the images you are comparing across both of the system’s monitors (only available with Dual Monitor option).

Compare Measurement Differences

Compare the properties of line or ROI measurements made on different Studies or Series. Compare the measurement differences between two Series of the same Study or compare the differences between two Studies.

3000067186691_A/881 * 2021-06-30

Philips



1. The first line or ROI to be compared.
2. The calculated difference (between 1 and 2) from lines or ROIs of two different series or studies.
3. The second line or ROI to be compared.

Compare is performed using any Viewer or Application that supports the Compare feature. The option is available from the context menu of the line and ROI annotations.

Compare Between Values

1. Load at least two series into a viewer or application.
2. Click the **Compare** button.
3. Draw the measurement on the first Study or Series using one of the line or ROI tools(number 1 in the illustration). See section “CT Common Tools” on page 141.
4. Draw a measurement on the second Study or Series (number 3 in the illustration).
5. Right-click the first measurement and select **%Difference**.
6. Repeat the previous step for the second measurement.

   The calculations appear at the bottom of the images (number 2 in the illustration).

Viewport Controls

Viewport Controls allow you to view and modify the image display parameters in the selected viewport. The Viewport Controls appear as blue, underlined text.

The displayed viewport controls are based on the active 2D, MPR, or Volume image.

When you click the control parameter, a selection box opens. There are three ways to change parameters within selection boxes. This applies to all viewport controls.

1. Click and highlight the current parameter in the selection box and type a new one over it.
2. Click the parameter selection box arrow buttons up or down to change the parameter.
3. Some parameters can be modified with a slider bar in the selection box.

NOTICE

CCA and other applications include some additional parameters and protocols in the viewport controls. Those viewport controls are described in the relevant sections.

3000067186 91_A/881 * 2021-06-30

2D Image Controls

These controls appear in the 2D image viewports. Access the controls in the upper and lower viewport corners.

Thickness

Use to change image thickness (upper right corner of viewport).

WL

Change the window level center (lower left corner of viewport) with this control.

WW

Adjust the window width (lower right corner of viewport).

Zoom

Zoom the image from 0.50 to 15 magnification (upper right corner of viewport, under the titles, if displayed).

Average

Average is the default rendering mode for 2D images (lower left corner of viewport).

Volume Image Controls

These controls appear in the volume image viewport. They can include some of the same controls as in 2D images, but do not include Average rendering mode.

Angle

All volume displays include the Angle parameter (located in the lower left corner of the main volume viewport).

Volume Rendering Mode Image Control

All rendering mode viewport controls are located in the lower right corner of volume viewports.

NOTICE

For more information on Volume Rendering Controls, see section “Image Rendering Mode” on page 232.

MIP

Set the maximum intensity projection. It is also associated with WL (window center), WW (window width).

3000067186 91_A/881 * 2021-06-30

VIP

Volume intensity projection is associated with WL and WW, but is also associated with the Width (of the ramp) parameters.

MinIP

Minimum intensity projection is associated with WL and WW, but also includes the Threshold HU (Hounsfield Units) parameter

SurfaceMIP

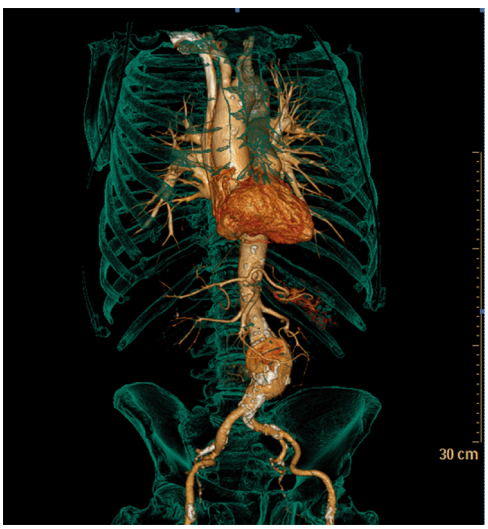
Surface maximum intensity projection is associated with WL and WW, but also includes Threshold HU (Hounsfield Units), and Distance parameters.

Vol. Rend.

Volume rendering is associated with Opacity (uses slider bar to modify parameter) and VR protocols.

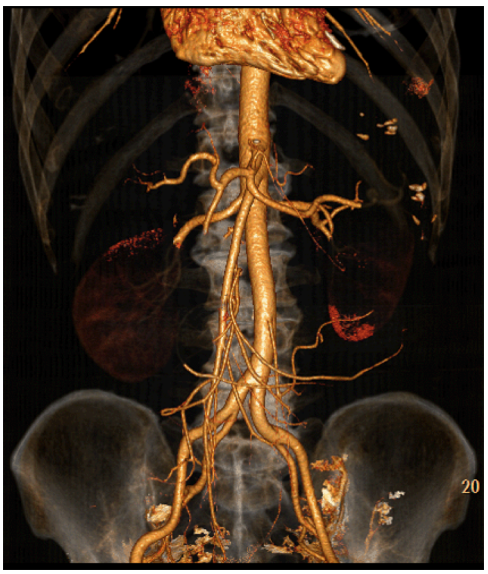
Glass View

In the Glass View mode a volumetric foreground anatomy is displayed against a glass-like transparency background. The background transparency shows all volumes removed by either bone removal or clipping.



Show Transparent

Click **Show Transparent** to view, in a semi-transparent mode, all volumetric anatomy that you have removed using various sculpting, clipping, or other tissue functions. (Bone is shown with less transparency than other tissues.)



Show Transparent Viewport Control

In the Show Transparent viewing mode, the Transparency viewport control appears in the lower left corner of the volume image viewport, as shown below, with the current transparency value.

The transparency value of the clipped tissues is adjustable in steps of 25 (0, 25, 50, 75, and 100). The value 0 represents least transparency and the value 100 represents maximum transparency.

- To change the current Transparency setting, click on the control title.
- Change the Transparency value using the slider control.

Bookmarks

While working with a patient study in a viewer or an application, you can use a bookmark at any time to “save the current status” of your work.

- You can save a bookmark at an intermediate point of your work process. This allows you to try a different procedure, and later, if desired, return to the saved point later by activating the saved bookmark.
- Or, you can save a bookmark and quit the current work session. In a later work session, you can activate the bookmark and continue working where you left off.

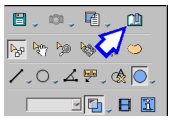
3000067186 91_A/881 * 2021-06-30

Philips

NOTICE

You can have a bookmark saved automatically. In the viewing applications page of Preferences activate “Save bookmark when exiting an application.”

Save Bookmark



As you work, you can continue saving bookmarks. When you save a bookmark, IntelliSpace Portal saves numerous existing parameters and data about your current work session. When you click Save bookmark (see section “CT Common Tools” on page 141) the Save Bookmark dialog box opens. The system will automatically supply a default name. You can accept the default name, or you can enter a new name for the bookmark in the Name: field, as shown above. (You must enter a new name for the bookmark or you will not get a mini image of it when you want to open it.)

When you click **OK**, the bookmark will be saved to your default device and will be associated with the patient study.

NOTICE

You can tell the system “Do Not Show This Dialog Again” to save time. The Save Bookmark window will not display again, and the bookmarks will be saved only by its sequential number.

Open Bookmark from Directory

You can activate saved Bookmarks from the Directory and from the application. The Directory version is shown below.

NOTICE

At any time, either in your current work session or at a later time, you can “open” a saved bookmark. In effect you "step into" a the application as it existed at the moment you saved the bookmark, and can continue working from that point on.

First select the study you want to work with. Existing bookmarks for the current application (or viewer) are shown as mini images.

If multiple bookmarks are available, click on the down arrow to view them. A right mouse menu offers options including Load Bookmark, Copy to Clipboard, and Email Link.

Click **Load Bookmark** to activate the function, and the IntelliSpace Portal will launch the patient study and display it in the condition it was in when you saved the bookmark.

3000067186691_A/881 * 2021-06-30

Philips

Open Bookmark from Application

Bookmarks in an application are accessed from the function tab. A right-click menu offers options including **Open**, **Copy Link to Clipboard**, and E**mail Link**. The **Copy Link to Clipboard** and **Email Link** functions allow you to share studies and work results with other users via email. Receiving users can access the link via the **Collaboration Viewer**.

NOTICE

At any time, either in your current work session or at a later time, you can open a saved bookmark. In effect you return to the application as it existed at the moment the **Bookmark** was saved.

Delete Bookmark

You can delete bookmarks via the right click context menu on States in the Patient Directory. You cannot delete a specific bookmark, you can only delete a State. See section “States” on page 260.

States

A state is a group of one or more bookmarks displayed in a window of mini images, each of which represents the “frozen” extent of your work at the time you created the bookmark.

A state is accessible from the Directory using the Bookmark icon. A state is the collection of all bookmarks that were created for one patient in one viewer or one application.

3000067186 91_A/881 * 2021-06-30

Philips

Magic Glass

The Magic Glass feature is available for all image viewports via the Magic Glass icon in the tool panel, or from the right click menu. You can have Magic Glass active on only one viewport at a time.

Magic Glass is an enhanced visualizing window that can be superimposed on top of your current image. The Magic Glass function improves your visualization of certain elements of the existing image, like calcium and stents, while maintaining optimal viewing parameters of the main viewport.

Magic Glass is a movable mini-window (default is 3cm square) which can be set with its own windowing, zooming, image enhancement and rendering parameters.

The Magic glass function is available on many tool panels (and via the right click menu). The Magic Glass can be easily moved, expanded, reduced, or hidden to best suit your needs:

- To move the Magic glass window, click on the edge of the Magic Glass frame and drag to the desired location.
- To reduce or expand the size of the window, drag the green frame.
- To change the viewing parameters inside the (active) Magic Glass window, use the viewport controls labeled MG.
- To change the viewing parameters outside the Magic Glass window, click outside the window and use the Common tools.

WARNING

All annotations drawn inside the Magic Glass ROI represent the underlying image and not the Magic Glass ROI. This means that all the centerlines, lumen lines, segmentation overlays, and cross-hairs will not change according to the image inside the Magic Glass ROI when zooming inside the Magic Glass ROI.

When measuring HU and SD values inside the Magic Glass ROI, be aware of the following: If the enhancement value inside the Magic Glass ROI is different from the enhancement of the underlying image, the HU and SD values are derived from the underlying image and not the Magic Glass ROI.

Magic Glass Presets

In the Comprehensive Cardiac Analysis and Cardiac Viewer applications the Magic Glass option includes factory presets and the ability to save user defined presets.

Right click inside the Magic Glass window to access the following functions:

Magic Glass

Activate Magic Glass feature using **Alt+M** on the keyboard.

3000067186 91_A/881 * 2021-06-30

Color Map

Show map of HU distribution in Magic Glass window.

Factory Defined

Access these presets: General Plaque, Non-calcified Plaque, Moderate Calcium, Stent, and Severe calcium.

User Defined

Access user presets, and these functions: "Save Preset As ..." and "Edit Presets List ..." (see section “Save Magic Glass User Presets” on page 262 and section “Edit Magic Glass User Presets” on page 262).

Save Magic Glass User Presets

Under the User defined option, click **Save preset as ...** to save the current settings to a preset. The following parameters are saved: Windowing, Enhancement, Rendering modes and parameters, Inverse, and Zoom. See also section “Magic Glass Presets” on page 261.

Edit Magic Glass User Presets

Under the User defined option, click **Edit presets list ...** if you want to rename or delete user defined preset(s). See also section “Magic Glass Presets” on page 261.

Common Keyboard Shortcuts

Function	Key	Function
Standard	<Ctrl>A	Select all in Directory.
	<Ctrl>C	Open Copy to dialog in Directory.
	Delete (Keyboard)	Delete selected items in Directory.
	F5	Refresh Directory.
Flip	<Ctrl>O	Rotate 180° vertical (flip orientation).
	O	Rotate 180° horizontal (flip orientation).
Orientation	A	Axial orientation.
	S	Sagittal orientation.
	C	Coronal orientation.
Annotation	D	Straight Line measurement
Film	F	Send image to film.
	<Shift>F	Send display to film.

3000067186 91_A/881 * 2021-06-30

Philips

Function	Key	Function
Report	R	Report images (sends images based on selection).
	<Shift>R	Report Display.
Save	<Shift>D	Save screen snapshot as ...
	<Shift>S	Save selected image(s) as ...
Viewing Tools	W	Windowing type-in dialog.
	T	Titles off/on.
	G	Grid off/on.
	I	Image parameters.
	M	Maximize/minimize the active viewport.
	B	Change background color for Volume Rendered image.
	X	Rotation Center Mode (slab) - the cursor will jump to the middle of view port.
	L	Link in Compare.
	<+>/<->	Increase/decrease zoom factor (zoom must be active).
	Page Up/Down	Scroll.
	Arrow Up/Down	Incremental scroll (functions like middle mouse button).
	Arrow Left/Right	Change thickness.
	Home/End	Scroll to beginning/end.
Windowing	0 through 9	Windowing presets (according to preferences).
Display Modes	<Shift> 1	Average.
	<Shift> 2	MIP.
	<Shift> 3	VIP.
	<Shift> 4	MinIP.
	<Shift> 5	Volume Rendering.
	<Shift> 6	Surface MIP.
Navigation	<Alt>N	Switch to Next series.

3000067186 91_A/881 * 2021-06-30

Philips

Function	Key	Function
Fusion Viewing	<Alt>Z	CCA - show hide overlay in Functional stage.
	P	In Dynamic Viewer function, switch between protocol presets available for currently loaded study. Press repeatedly if multiple presets exist.

Common Mouse Functions

Mouse Button	Function
Left	According to the operation mode.
Middle	Roll: scroll. Drag: windowing.
Right	Click: open context menu. Drag & Drop: swap viewports.
Left + middle	Pan.
Left + Right	Swivel.
Middle + Right	Zoom.
<Shift> Left mouse	Change thickness.
<Shift> Middle mouse	Change window width.
<Ctrl> Middle mouse	Change window center.
<Alt> Middle mouse	Change Opacity of Volume rendering.
<Ctrl> Left mouse	When using the mouse in the non-default mode (default is usually scroll or swivel) hold down control to activate the default mode.
<Ctrl> Pan + Left mouse	Vertically restricted pan (must be in pan mode).
<Alt> Pan + Left mouse	Horizontally restricted pan (must be in pan mode).
<Ctrl> + Left mouse	When selecting items with the left mouse, hold down <Ctrl> to Add/Remove items.
<Shift> + Left mouse	Select Range.

3000067186691_A/881 * 2021-06-30

Philips