Chapter 3

Building Node Network

Learning Objectives

After completing this chapter, you will be able to:

- *Add a node in the Node Graph panel*
- *Connect a node with other nodes*
- *Select, edit, replace, rename, group, and clone nodes*
- *Enhance the workflow using the Precomp, Backdrop, and Dot nodes*
- *Premultiply images using the Premult node*
- *Understand Properties Bin and properties panels*
- *Understand common node controls in the properties panels*
- *Use the Image nodes*
- *Merge images using the Merge node*

INTRODUCTION

Nodes are the building blocks of a NukeX script. A node in NukeX is represented by a button like shape with its name displayed on it. NukeX has a node-based interface in which complex processes are built up by connecting nodes to form a tree (or flow) called process tree, refer to Figure 3-1. A process tree (node network) is an efficient way to manipulate images to create the desired effects. You can start a NukeX script by importing images by using the **Read** nodes, connecting them with other process nodes, and then finally rendering the scene to get the output.

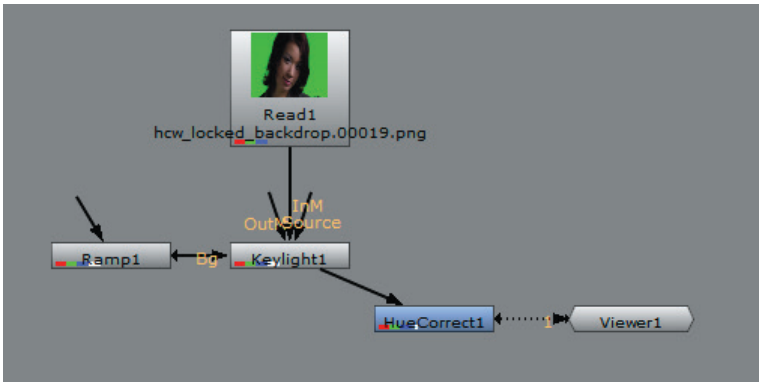


Figure 3-1 *A simple process tree*

In this chapter, you will learn to connect, add, select, replace, edit, clone, de-clone, group, and rename the nodes. Also, you will create the first composition using the **Merge** node.

INSERTING AND MANIPULATING NODES

You can insert a node in the **Node Graph** panel by using the **Nodes** toolbar, the TAB menu, the shortcut menu, or shortcut keys. All these methods are discussed next.

Inserting Nodes Using the Nodes Toolbar

To insert a node using the **Nodes** toolbar, choose the required category button from it and then choose the node from the menu displayed. Figure 3-2 shows the nodes available in the **Image** menu. The chosen node will be inserted in the **Node Graph** panel, refer to Figure 3-3. You will notice that whenever you insert a new node in the **Node Graph** panel, its properties panel will be displayed in the **Properties Bin**, refer to Figure 3-4.



Tip: *You can insert the last chosen node from the* ***Nodes*** *toolbar by clicking the category button with the middle-mouse button.*

To view the output of the inserted node in the **Viewer#** panel, make sure the node is selected in the **Node Graph** panel. Next, press 1; the Output connector of the selected node will be connected to the port **1** of the **Viewer#** node, refer to Figure 3-5.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

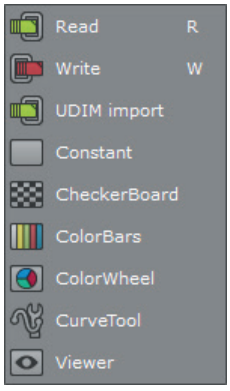


Figure 3-2 *The* ***Image*** *menu displayed on choosing the* ***Image*** *button*

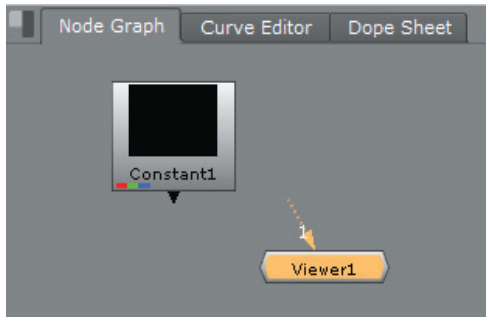


Figure 3-3 *The* ***Constant1*** *node inserted in the* ***Node Graph*** *panel on choosing* ***Constant*** *from the* ***Image*** *menu*

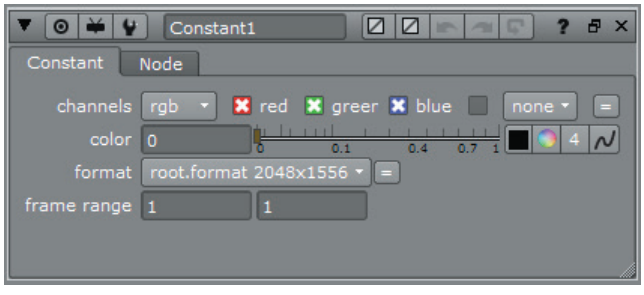


Figure 3-4 *The* ***Constant1*** *node properties panel displayed on inserting the* ***Constant1*** *node in the* ***Node Graph*** *panel*

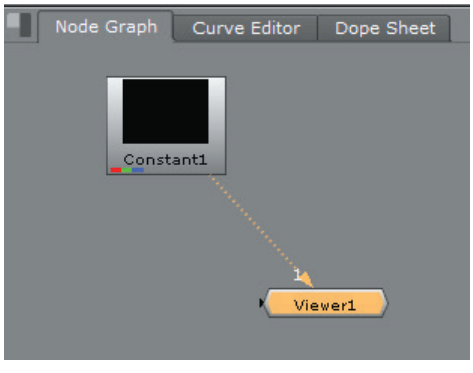


Figure 3-5 *The* ***Constant1*** *node connected to the port* ***1*** *of the* ***Viewer1*** *node*

Inserting Nodes Using the Shortcut Menu

To insert a node using the shortcut menu, right-click on the **Node Graph** panel; a shortcut menu will be displayed. Choose the required category from it; a cascading menu will be displayed, refer to Figure 3-6. Choose the required node from the cascading menu to insert it in the **Node Graph** panel.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

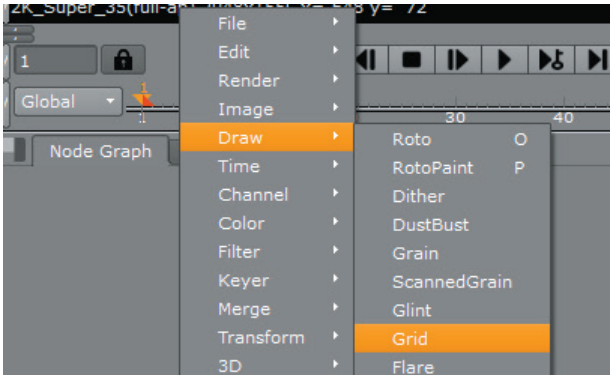


Figure 3-6 *Partial view of the* ***Draw*** *cascading menu*

Note

If a node is selected in the ***Node Graph*** *panel, then on adding a new node, it will be automatically connected to the selected node. Else, it will remain as a free floating node.*

Inserting Nodes Using the TAB Menu

To insert a node using the TAB menu, hover the cursor over the **Node Graph** panel and then press TAB; the TAB menu will be displayed. Next, type the first few characters of the name of the node to be inserted in the TAB field; NukeX will display all the nodes whose names begin with the typed characters, refer to Figure 3-7. You can use the UP or DOWN arrow key to navigate through the menu. Next, select the required node in the TAB menu and then press ENTER; the selected node will be inserted in the **Node Graph** panel.

Tip: *To reinsert the last node from the TAB menu, press TAB followed by ENTER.*

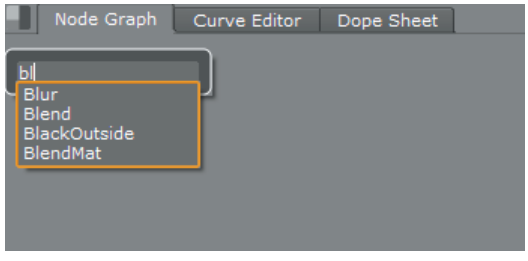


Figure 3-7 *The nodes displayed in the TAB menu on typing* ***bl*** *in the TAB menu field*

Inserting Nodes Using the Shortcut Keys

To insert a node using the shortcut keys, click on the **Node Graph** panel to make it active and then press the shortcut key corresponding to the node that you need to add. The shortcut keys corresponding to the nodes are displayed in the category menus. These menus are displayed when you choose the category button from the **Nodes** toolbar. Figure 3-2 shows the shortcut keys R and W corresponding to the **Read** and **Write** nodes, respectively.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

WORKING WITH NODE CONNECTIONS

Once you add nodes to the **Node Graph** panel, you need to establish the connections between nodes to form a network. Most of the nodes have the Input, Output, and Mask connectors. These connectors determine the order in which the data stream is passed through the nodes. Figure 3-8 shows the connectors on the **Blur1** node. Some nodes such as **Merge** has two input connectors: **A** and **B**. Figure 3-9 shows the connectors of a **Merge** node.

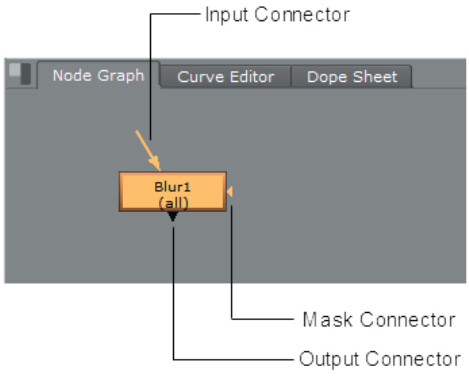


Figure 3-8 *Connectors on a* ***Blur1*** *node*

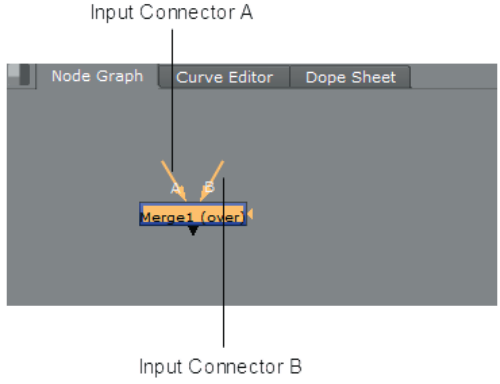


Figure 3-9 *The* ***Merge1*** *node with two input connectors*

Disconnecting Nodes

To disconnect a node, press and hold the left mouse button on the head or tail of the connecting arrow and then drag the cursor to an empty area of the **Node Graph** panel. Alternatively, you can select the node and then press CTRL+D.

Reconnecting Nodes

To reconnect a node, press and hold the left mouse button on the head or tail of the connecting arrow of the node that you want to connect. Now, drag the cursor to the center of the node to which you need to make the connection and then release the left mouse button; a connection will be established between the two nodes. Alternatively, select the node (lower in the hierarchy) that you want to connect and then press and hold SHIFT. Next, select the node that you need to connect and then press Y; a connection will be established between the two nodes.

Extracting Nodes

To extract a node, select the node that you need to extract from the node network and then choose **Edit > Extract** from the menu bar; the selected node will be extracted from the node network and will be placed as a free floating node in the **Node Graph** panel. Alternatively, you can press CTRL+SHIFT+X to extract the node. Figure 3-10 shows the **Grade2** node extracted from the node network.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Swapping Inputs of Nodes

Some nodes like **Merge** have two inputs: **A** and **B**. To swap the **A** input with the **B** input, make sure that the **Merge** node is selected in the **Node Graph** panel and then choose **Edit > Node > Swap A - B** from the menu bar. Alternatively, you can press SHIFT+X to swap the inputs. Figures 3-11 and 3-12 show the default and swapped inputs, respectively.

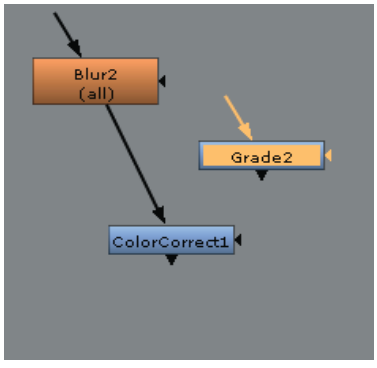


Figure 3-10 *The extracted* ***Grade2*** *node*

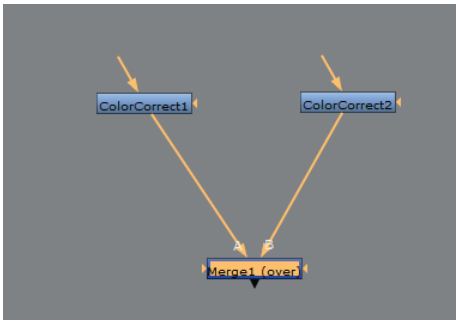


Figure 3-11 *The default* ***A*** *and* ***B*** *inputs*

Starting a New Branch from a Node

To create a new branch from a node, select the node from which you want to create a new branch. Next, press and hold SHIFT and then add a new node using the **Nodes** toolbar, TAB menu, shortcut menu, or the shortcut key. Figure 3-13 shows two branches originating from the **Blur3** node.

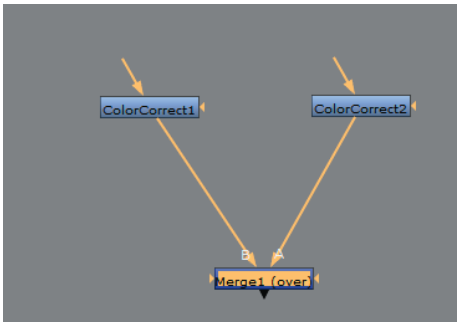


Figure 3-12 *The swapped inputs*

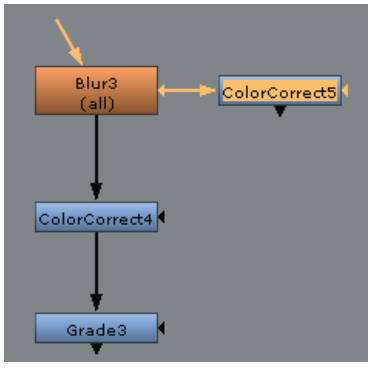


Figure 3-13 *The branches originating from the* ***Blur3*** *node*

Duplicating the Connecting Arrow

To duplicate the connecting arrow, press and hold SHIFT and then drag the head of the output connecting arrow; a duplicate connection pipe will be displayed. Next, drop the pipe onto the node that you want to connect. This will duplicate the connecting arrow without affecting the original connection. Figure 3-14 shows a duplicate output arrow connected to the **Grade2** node from the **Blur2** node.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

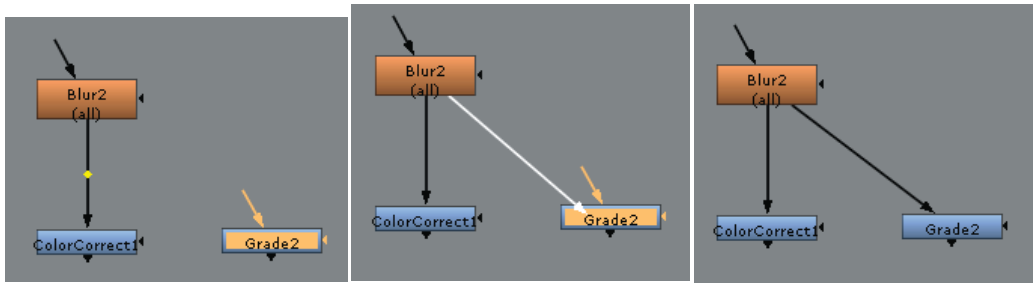


Figure 3-14 *Duplicating a connecting arrow*

SELECTING NODES

When you select nodes in the **Node Graph** panel, the selected nodes are displayed in a light yellow color. Various methods of selecting the nodes are discussed next.

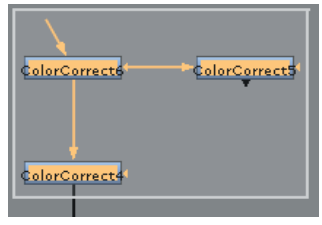


Figure 3-15 *Selecting the nodes by drawing a marquee selection*

To select a single node, click on the node once. To select multiple nodes, press and hold SHIFT and then click on each node that you need to select. Alternatively, you can draw a marquee selection around the nodes to be selected. All the nodes that lie within the marquee will be selected, as shown in Figure 3-15. To invert a selection, choose **Edit > Invert Selection** from the menu bar.

To select all upstream nodes, press and hold CTRL while dragging a node; all upstream nodes that feed data to the dragged node will be selected, refer to Figure 3-16. Next, click once in the empty workspace to clear the selection.

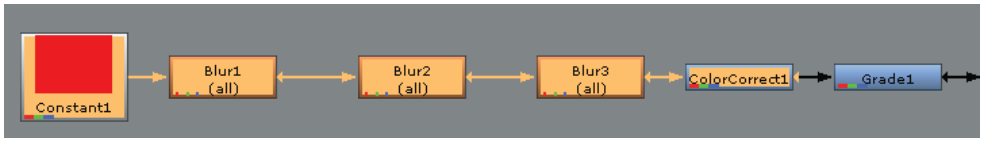


Figure 3-16 *All upstream nodes selected on dragging the* ***ColorCorrect1*** *node with CTRL held down*

To select all downstream and upstream nodes connected to a node, click once on the node and then choose **Edit > Select Connected Nodes** from the menu bar. Alternatively, you can press CTRL+ALT+A to select all nodes connected to the selected node, refer to Figure 3-17. To select all nodes, choose **Edit > Select All** from the menu bar. Alternatively, press CTRL+A to select all nodes in a script.

Tip: *Press and hold CTRL+SHIFT and then click to select one or more nodes without clearing the selection made earlier.*

To select a node by its name, choose **Edit > Search** from the menu bar; the **Nuke** dialog box will be displayed, as shown in Figure 3-18. Type the name of the node in the search field and

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

then choose the **OK** button; the node will be selected in the **Node Graph** panel. Also, the selected node will be centered in the visible workspace area. You can also use wild cards to select nodes by the name. You can use asterisks for multiple alphanumeric search strings and question marks for representing a single character search.

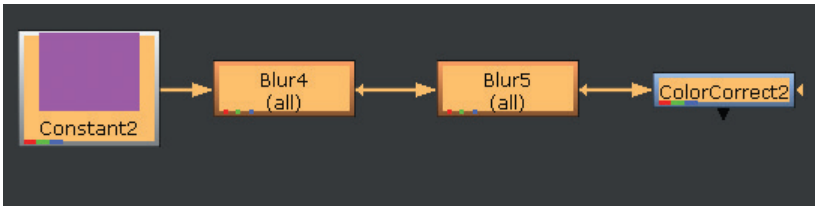


Figure 3-17 *All upstream and downstream nodes connected to the* ***Blur5*** *node selected in the* ***Node Graph*** *panel*

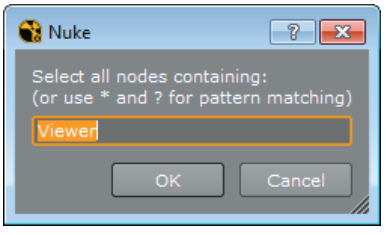


Figure 3-18 *The* ***Nuke*** *dialog box*

You can also select nodes based on their color, class, and label. For example, to select nodes based on the color, select a node and then choose **Edit > Select Similar > Color** from the menu bar; all nodes having the same color will be selected in the **Node Graph** panel.

REPLACING NODES

To replace a node, select the node in the **Node Graph** panel that you need to replace. Next, press and hold CTRL and create new node using the **Nodes** toolbar, shortcut menu, or the TAB menu; the selected node will be replaced by the new node.



Tip: *To replace a node with the TAB menu, select the node in the* ***Node Graph*** *panel that you need to replace. Next, press TAB and then type first few characters of the name of the node. Next, press and hold CTRL and then select the new node from the TAB menu. If you are using a hotkey, you cannot replace nodes using this method.*

RENAMING NODES

To rename a node, double-click on the node; its properties panel will be displayed. Next, in the properties panel, type the new name in the **name** field and then press ENTER, refer to Figures 3-19 and 3-20. Alternatively, you can rename a node by selecting it in the **Node Graph** panel and then pressing N; a text box will appear on the node. Next, type the new name in the text box and then press ENTER to rename the node.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

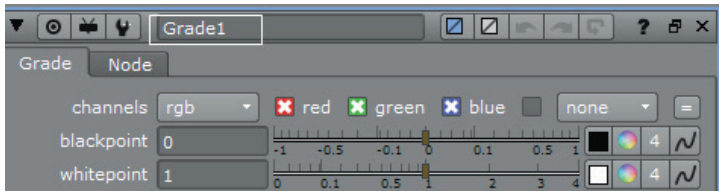


Figure 3-19 *The name of the node displayed in the* ***name*** *field of the* ***Grade1*** *node properties panel*

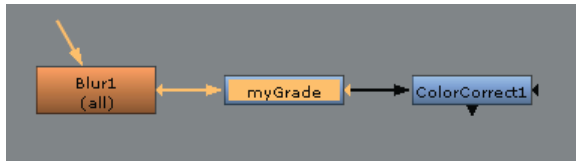


Figure 3-20 *The* ***Grade1*** *node renamed to* ***myGrade***

EDITING NODES

Standard shortcut keys (*CTRL+C*, *CTRL+V*, and *CTRL+X*) are used to perform editing operations on the nodes. To copy a node in the memory, select the node and then choose **Edit > Copy** from the menu bar. Next, click on the empty area of the workspace to deselect any selected node and then choose **Edit > Paste** from the menu bar; an exact copy of the selected node is created in the **Node Graph** panel.



Note

When you paste nodes, NukeX automatically connects them to the node that is selected before the paste command.

You can also copy nodes to the files. To do so, select the node that you need to copy in the **Node Graph** panel. Next, choose **File > Export Nodes As Script** from the menu bar; the **Export Nodes As Script** dialog box will be displayed, refer to Figure 3-21.

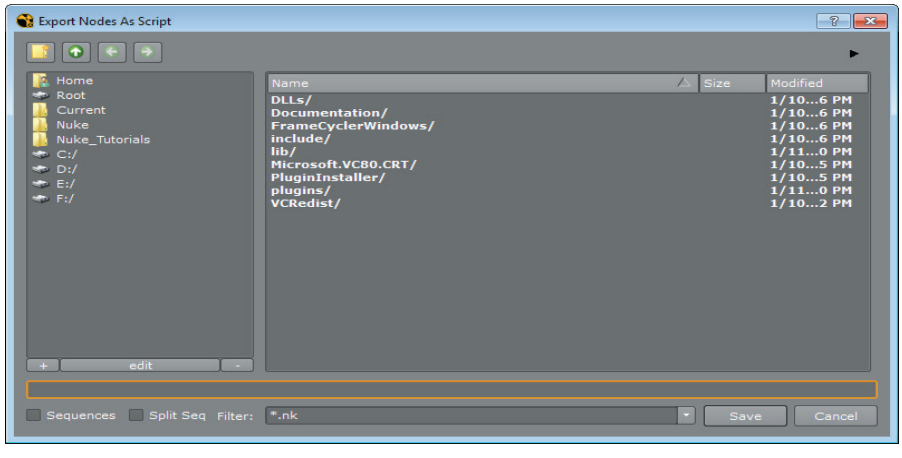


Figure 3-21 *The* ***Export Nodes As Script*** *dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Next, navigate to the folder where you want to save the script and then enter name of the script in the pathname field. Next, choose the **Save** button. NukeX will save the selected node as a script with the specified name. Start Notepad and open the saved script in it to view its content. Figure 3-22 shows the content of the **ColorCorrect** node script opened in Notepad application.

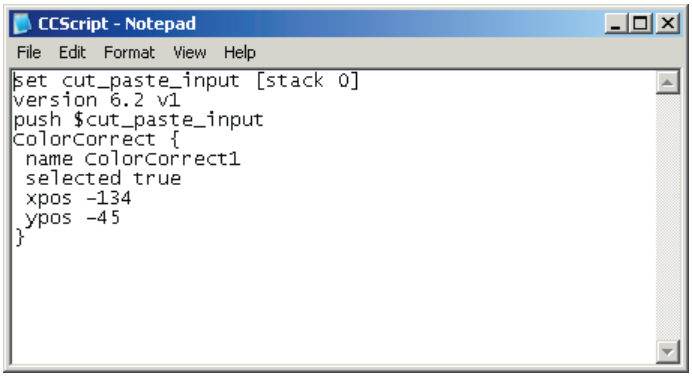


Figure 3-22 *The* ***Color Correct*** *node script displayed in Notepad*

You can import the saved script files in NukeX as a node. To do so, choose **File > Import Script** from the menu bar; the **Import Script** dialog box will be displayed, as shown in Figure 3-23. Next, navigate to the folder where you have saved the script and then select the script file. Next, choose the **Open** button from the dialog box; the selected script file will be imported in the **Node Graph** panel.

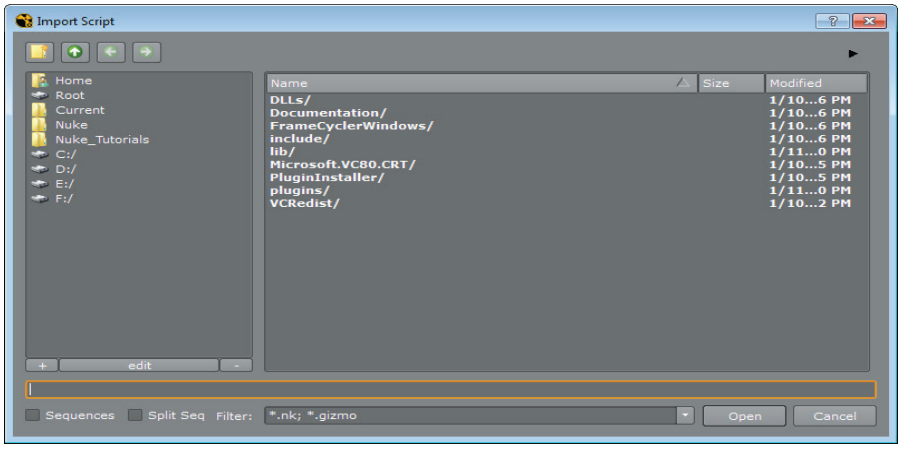


Figure 3-23 *The* ***Import Script*** *dialog box*

To cut the node from the **Node Graph** panel, select the node and then choose **Edit > Cut** from the menu bar; NukeX will remove the selected node from the script and write it to the memory. Next, select the node that you want to connect with the node in the memory. Choose **Edit > Paste** from the menu bar; NukeX will insert nodes to the **Node Graph** panel from the memory.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

To create a duplicate node, select the node in the **Node Graph** panel and then choose **Edit > Duplicate** from the menu bar or press ALT+C. A duplicate of the selected node will be created in the **Node Graph** panel. To delete a node, select the node in the **Node Graph** panel and then choose **Edit > Erase** from the menu bar or press DELETE; the selected node will be removed from the **Node Graph** panel. To disable or enable a node, select the node that you want to disable or enable in the **Node Graph** panel and then choose **Edit > Node > Disable/ Enable** from the menu bar.



Tip: *To toggle between enable and disable modes of a node, press D.*

CLONING NODES

Clones are used to maintain consistency throughout the script. Cloned nodes inherit the values of their parent nodes maintaining an active link with them.



Note

You cannot clone gizmos used in NukeX even if they are user created or built-in.

To clone a node, select the node in the **Node Graph** panel. Next, choose **Edit > Clone** from the menu bar or press ALT+K; NukeX will create a clone of the selected node and maintain an active link with the parent node. A clone indicator (C) will be displayed on both the nodes and they also share the same name. To declone nodes, select them and then choose **Edit > Declone** from the menu bar or press ALT+SHIFT+K; NukeX will declone the selected nodes and remove the active link. Two nodes can be forced to be clones in a script. To do so, select first node. Next, press and hold SHIFT and then select the second node. Next, choose **Edit > Force Clone** from the menu bar or press CTRL+ALT+SHIFT+K; NukeX will make the first node as forced clone of the second node.

GROUPING NODES

You can group (visually organize) nodes in the **Node Graph** panel using the **Backdrop** node or the **Group** node. The **Backdrop** node is used to visually organize areas of the **Node Graph** panel. The **Group** node is used to group several nodes into a single node, and the group node thus created acts as a container for these nodes. The **Backdrop** and **Group** nodes are discussed next.

Grouping Nodes Using the Backdrop Node

To group nodes using the **Backdrop** node in the **Node Graph** panel, choose the **Other** button from the **Nodes** toolbar; the **Other** menu will be displayed. Next, choose **Backdrop** from the menu; the **BackdropNode#** will be inserted in the **Node Graph** panel, refer to Figure 3-24. In the **Node Graph** panel, drag the triangle on the lower right corner of the **Backdrop** node to increase its size, as required. Next, press and hold the left mouse button over the title of the box and then drag the node and slip it under the nodes that you need to visually organize, refer to Figure 3-25. Next, click on the empty area of the **Node Graph** panel and then drag the title of the **Backdrop#** node. You will notice that nodes grouped using the **Backdrop** node move together with it.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

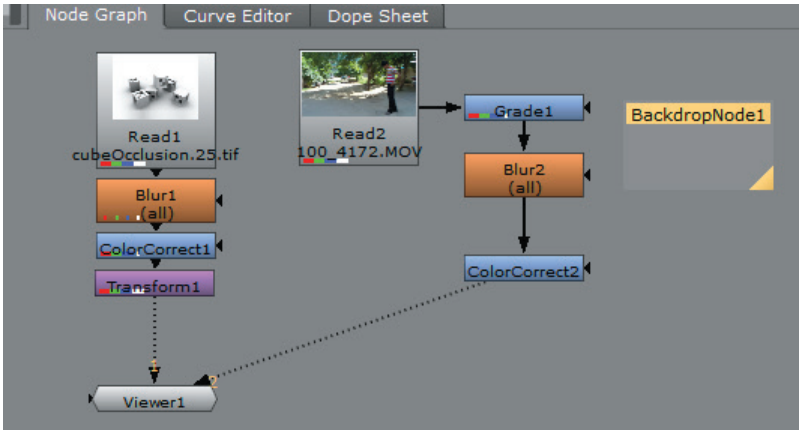


***Figure 3-24** The **BackdropNode1** node inserted in the **Node Graph** panel*

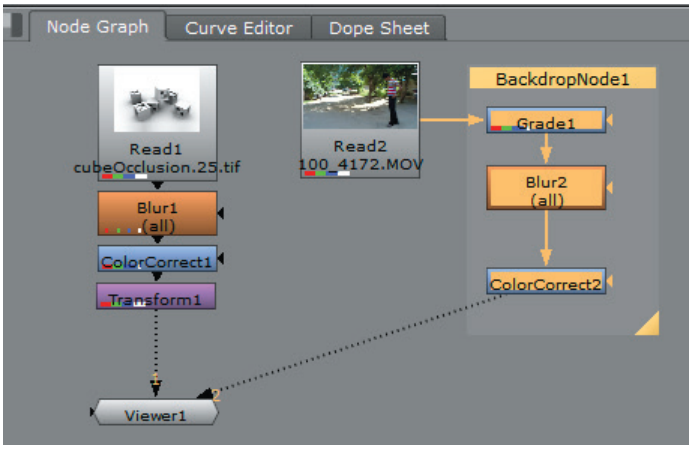


***Figure 3-25** The **Grade1**, **Blur2**, and **ColorCorrect2** nodes visually grouped using the **BackdropNode1** node*

To display the properties panel of the **Backdrop** node, double-click on it in the **Node Graph** panel. To change the color of the **Backdrop** node, choose the **tile_color** button from the **BackdropNode#** properties panel; the **BackdropNode#.title_color** window will be displayed. Next, select the color as required. To change the name of the **Backdrop** node, enter the new name in the **name** field of the **Backdrop** node properties panel. To delete the **Backdrop** node and the nodes it contains, click once on the title of the node and then press DELETE. To delete only the **Backdrop** node, CTRL+click on the title of the node and then press DELETE.

Grouping Nodes Using a Group Node

To group nodes using the **Group** node, select the nodes that you need to group, refer to Figure 3-26. If you want to replace the original nodes with the **Group** node, choose **Edit > Node > Group > Collapse to Group** from the menu bar or press CTRL+G; the **Group#** node will be inserted in the **Node Graph** panel. Also, a new panel will be displayed with the name **Group# Node Graph**, refer to Figure 3-27.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

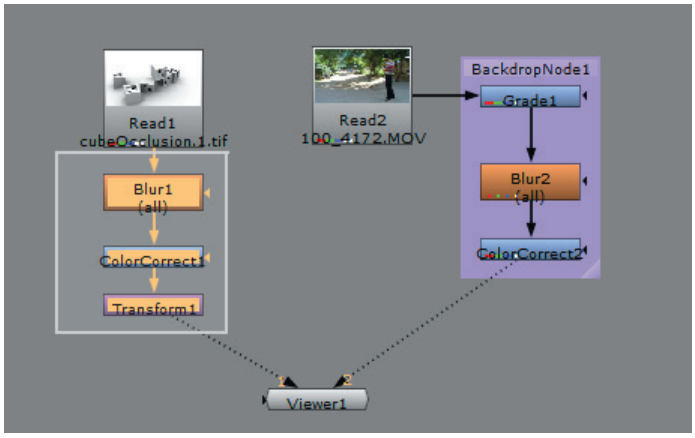


Figure 3-26 *Selecting the nodes in the* ***Node Graph*** *panel*

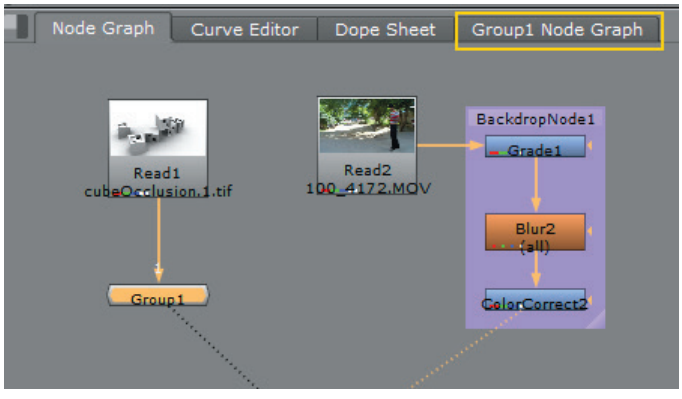


Figure 3-27 *The* ***Group1*** *node and the* ***Group1 Node Graph*** *panel*

To expand (ungroup) a group, select the **Group#** node in the **Node Graph** panel and then choose **Edit > Node > Group > Expand Group** from the menu bar or press CTRL+ALT+G. To open the **Node Graph** panel for the group, select the **Group#** node in the **Node Graph** panel and then choose **Edit > Node > Group > Open Group Node Graph** from the menu bar or press CTRL+ENTER, refer to Figure 3-28. Alternatively, you can press the show **S** button in the **Group#** node properties panel.

To keep the selected nodes in the **Node Graph** panel, choose **Edit > Node > Group > Copy Node To Group** from the menu bar or press CTRL+ALT+SHIFT+G; a free floating **Group#** node will be created in the **Node Graph** panel along with a separate **Group# Node Graph** panel, refer to Figure 3-29.

ENHANCING WORKFLOW

Some nodes in the **Other** category of the **Nodes** toolbar are used to enhance the workflow. The **StickyNote** node is used to add notes to the **Node Graph** panel. The notes can be text or HTML markup. Generally, they are used as annotations to other nodes in the **Node Graph**

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

panel. You can use the **Dot** node to make bends in the connecting pipes. It helps you in organizing your script better. The **PostageStamp** node is used to display a thumbnail render of the output at the current frame. This node does not alter the data stream. The nodes used to enhance the workflow are discussed next.

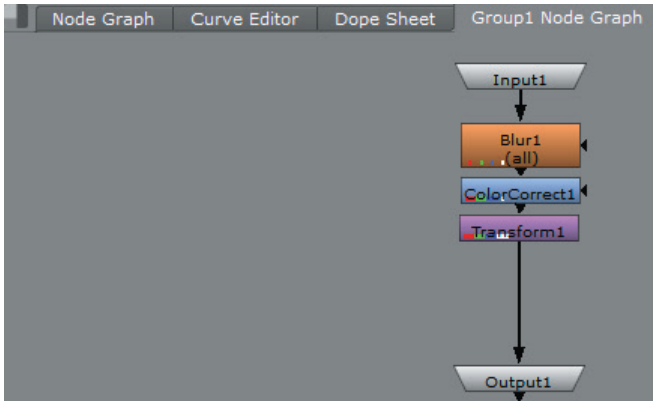


***Figure 3-28** The **Group1 Node Graph** panel displayed*

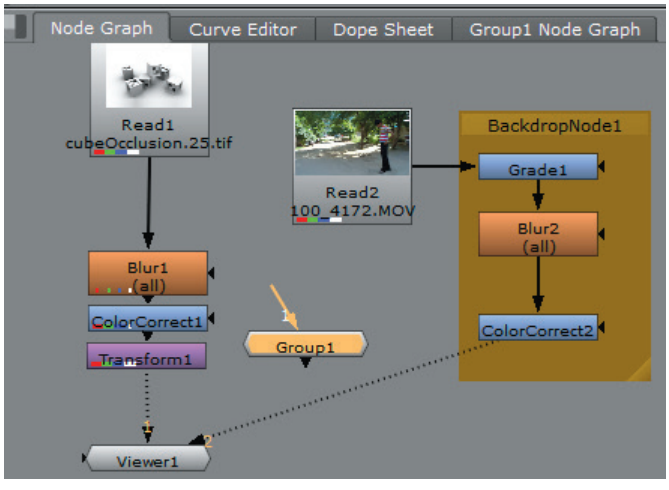


***Figure 3-29** A floating **Group1** node created*

Using the StickyNote Node

The **StickyNote** node is used to add annotation to nodes in the **Node Graph** panel. To add a note to the **Node Graph** panel, click on the area of the **Node Graph** panel where you need to add a note. Next, choose the **Other** button from the **Nodes** toolbar; the **Other** menu will be displayed. Now, choose **StickyNote** from the menu; the note will be displayed in the **Node Graph** panel, as shown in Figure 3-30. To change the note text, enter new text in the **label** field of the **StickyNote#** node properties panel. You can also enter HTML markup such as * your_note_text_here * for note to appear in bold or * your_note_text_here * for note to appear in italics in the **Node Graph** panel. You can also add an image to your note. To do so, use ** HTML markup.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Note

The commonly used image formats can be used for the logo but using the png format is highly recommended. Also, you need to save the image files in the plug-in path directory (C:\Program Files\Nuke7.0v4\plugins) in order to make images appear on the notes.

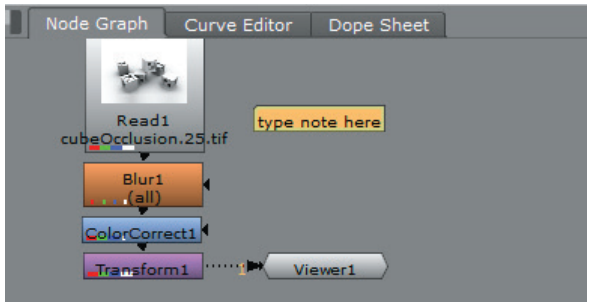


Figure 3-30 *A note displayed in the* ***Node Graph*** *panel*

Using the Precomp Node

The **Precomp** node is similar to the **Group** node. However, its contents are stored in an separate NukeX script file. It helps in breaking a large script into smaller process trees. The **Precomp** node enhances the workflow and speeds up the rendering process. In this case, NukeX has to process a single image input instead of all nodes used to create the precomp. This node enhances the workflow where one user can work on the main comp while other can work on sub-comps that can be edited independent of the main comp.

To add a **Precomp** node, select the nodes that you want to include in the separate script and then choose the **Other** button from the **Nodes** toolbar; the **Other** menu will be displayed. Next, choose **Precomp** from the menu; the **Precomp Nodes** dialog box will be displayed, as shown in Figure 3-31.

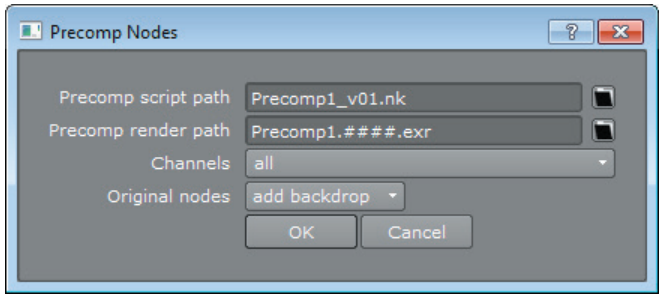


Figure 3-31 *The* ***Precomp Nodes*** *dialog box*

Note

If you do not select a node while creating a ***Precomp*** *node, the* ***Precomp Nodes*** *dialog box will not be displayed. However, you can add link to an existing script file to the empty* ***Precomp*** *node by using the* ***file*** *parameter from the* ***Precomp#*** *node properties panel.*

In this dialog box, click on the folder icon corresponding to the **Precomp script path** parameter; the **panelNode: Replace Precomp#_v01.nk** dialog box will be displayed. Next,

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

browse to the location where you want to save the precomp *.nk* script. Enter a name in the pathname name field for the precomp and then choose the **Open** button. Now, click on the folder icon corresponding to the **Precomp render path** parameter; the **panelNode: Replace Precomp#.####.exr** dialog box will be displayed. In this dialog box, browse to the location where you need to save the render of the precomp. Enter a name in the pathname field for the render file and then choose the **Open** button.



Note

By default, the precomp script is saved at the location where the main script exists. If you have not saved the main script, the precomp script is saved in the main directory.

Select the channels that you need to include in the precomp from the **Channels** drop-down of the **Precomp Nodes** dialog box. Select the desired option from the **Original nodes** drop-down. The options in this drop-down are used to determine the behavior of the selected nodes. Select the **add backdrop** option from this drop-down to add a backdrop behind the precomped nodes. Select the **delete** option from the drop-down to delete the precomped nodes. Select the **no change** option from the drop-down to retain its existing state.

Now, choose the **delete** option from the **Original nodes** drop-down and then choose the **OK** button from the **Precomp Nodes** dialog box; the **Group Output** dialog box will be displayed. In this dialog box, select the node that will work as output. Next, choose the **OK** button to close the dialog box. NukeX automatically creates a **Write** and an **Output** node in the precomp. On rendering, if NukeX cannot find any **Write** or **Output** node in the precomp, it produces an error in the precomp.



Note

1. You should render the image in the .exr format as it is capable of storing hash value. NukeX uses hash value to match the changes between the incoming node tree and the rendered file. If hash values do not match, NukeX notifies you about the mismatch. If you are using any other format than .exr, you will not get this notification and there is a risk of using out-of-date render file.

2. It is recommended that you set the project settings in the main comp. The precomp uses project settings of the main comp. If you open the precomp in a separate instance of NukeX and change its project settings, the change will have no effect on the main comp when it is loaded into the main comp.



Tip: *To display hash value of a node, select the node in the* ***Node Graph*** *panel and press I; a window with information about node, including the hash value, will be displayed, refer to Figure 3-32.*

Rendering a Precomp Node

Create a **Precomp** node as discussed earlier. In the **Precomp** tab of the **Precomp#** node properties panel, choose the **Render** button; the **Execute** dialog box will be displayed, as shown in Figure 3-33. Specify the frame range that you need to render in the **frame_range_string** field located next to the **Frame range** drop-down and then choose the **OK** button to close the dialog box. On doing so, the **Progress** window will be displayed showing the rendering

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

progress. It will automatically close once the rendering process completes. When the rendering process finishes, the **read file for output** check box in the **Precomp** tab of the **Precomp** node properties panel will be automatically selected. As a result, NukeX will read image sequence rather than calculating output of the precomp. You will also notice that the **Precomp#** node in the **Node Graph** panel has turned green. Also, the label **(Read)** will be displayed on it, refer to Figure 3-34. In case NukeX generates error on opening the rendered output, you can select the response from the **missing frames** drop-down. This drop-down has four options, namely **error**, **black**, **checkerboard**, and **read input**.

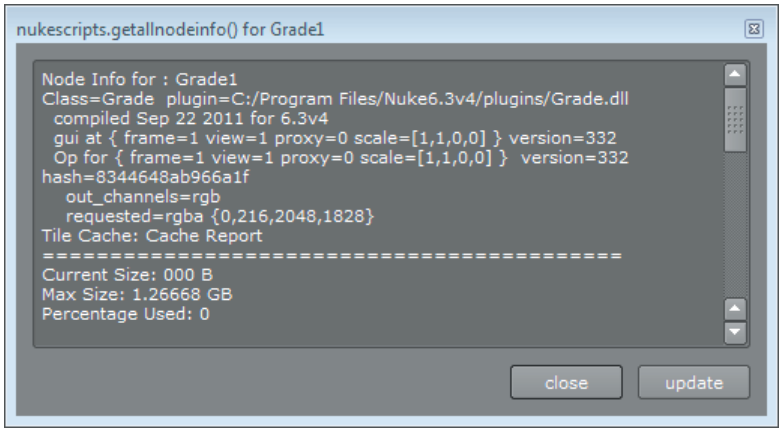


Figure 3-32 *A window displaying hash value of the* ***Grade1*** *node*

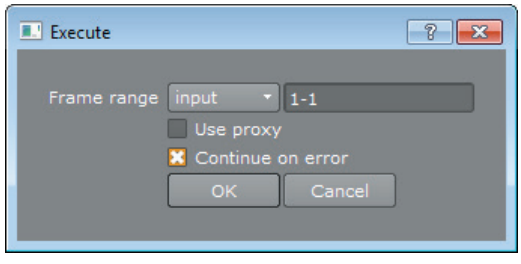


Figure 3-33 *The* ***Execute*** *dialog box*

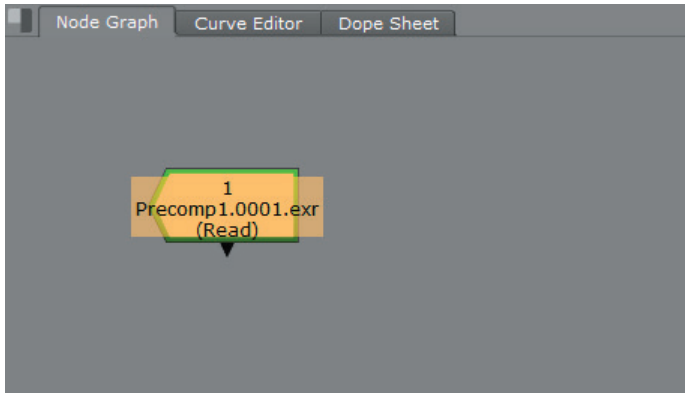


Figure 3-34 *The green colored* ***Precomp1*** *node with the* ***Read*** *label on it*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Viewing and Editing a Precomp Script

To view and edit a precomp script, choose the **Open** button in the **Precomp** node properties panel. On doing so, a new NukeX session will be launched and the precomp will be loaded in it. If you are using version numbers in the **Write** node, make sure to choose **Edit > Node > Filename > Version Up** from the menu bar; the version number of the file will be increased by one. For example, the *myPrecomp_v01_###.exr* will change to *myPrecomp_v02_###.exr*. Now render the **Write** node. Next, choose **File > Save New Version** from the menu bar or press ALT+SHIFT+S; the script will be saved with a new version. Fox example, the *myPrecomp_v01.nk* script will change to *myPrecomp_v02.nk*.

Reloading a Revised Precomp Script

To reload a revised precomp script, load the main comp. Next, in the **Precomp** tab of the **Precomp#** node properties panel, make sure the name of the script in the file field matches with the current name of the precomp script. Next, choose the **Reload** button to reload the revised precomp into the main comp.

Using the Dot Node

The **Dot** node is used to create bends in the arrows connecting nodes. This node helps in organizing the node network in the **Node Graph** panel. To create a bend in the connecting arrows, select the node after which you need the **Dot** node to appear and then choose **Dot** from the **Other** menu; the **Dot#** node will be displayed in the **Node Graph** panel, refer to Figure 3-35. Next, drag the **Dot#** node as required.

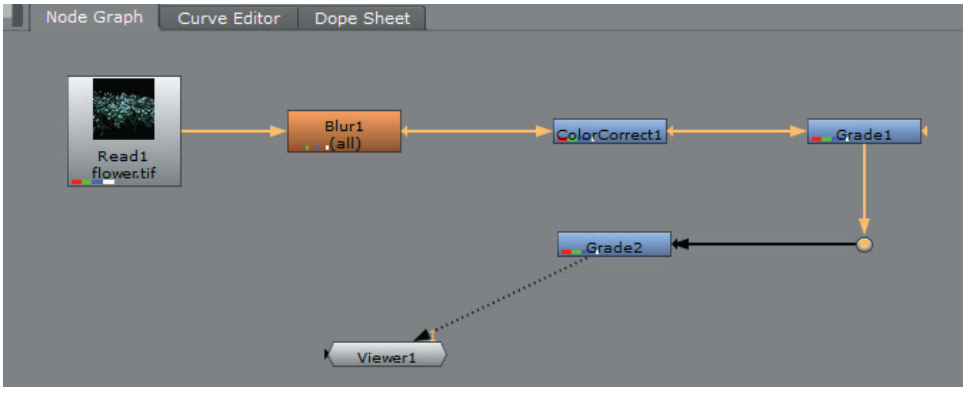


Figure 3-35 *The connecting pipe between the* ***Grade1*** *and* ***Write1*** *nodes bent using the* ***Dot*** *node*



Tip

1. You can use the . (period) key to add a ***Dot*** *node.*

2. You can also add a ***Dot*** *node to an existing connection by using the CTRL key. To do so, press and hold the CTRL key; a yellow diamond shape will appear on the connecting arrow. Next, click on the shape to insert a* ***Dot*** *node on the connecting arrow.*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

INDICATORS ON NODE

NukeX displays some indicators on nodes depending on the operation you are performing on them. These indicators are discussed next.



The node's effect is masked.



The wide rectangles indicate the channels processed by the node. The narrow rectangles indicate the channels that are not altered by the node.



The node is disabled. The data stream will pass through the node unaffected.



The node is disabled using an expression.



The node is cloned.



The fields of the node are animated over time.



One or more of the node fields are being affected by an expression.



One or more views in the node's controls are split off.



A multi-view project is opened and you have split off views in the node's control.



The node is not at its full effect. The node's effect is mixed with the original.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

WORKING WITH PROPERTIES PANELS

The properties panels are used to define the parameters of the nodes. When you add a node to the **Node Graph** panel, its properties panel automatically appears in the **Properties Bin**. Figure 3-36 shows the **Blur1** node properties panel. Many properties panels contain several tabs, as shown in Figure 3-37.

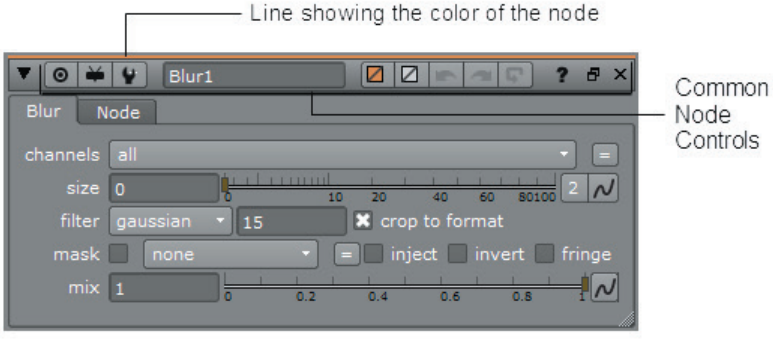


Figure 3-36 *The **Blur1** node properties panel*

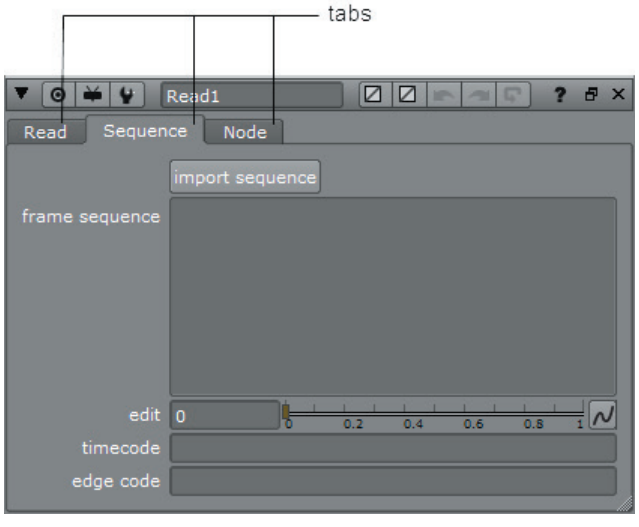


Figure 3-37 *The **Read1** node properties panel with tabs*

If the properties panel is not open for a node, you can use any of the following methods to open it:

- Double-click on the node in the **Node Graph** panel.
- Press and hold CTRL and then click on the node in the **Node Graph** panel.
- Select the node in the **Node Graph** panel and then press ENTER.
- Press and hold CTRL+ALT and then click on the node to open a properties panel in a floating window, refer to Figure 3-38.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

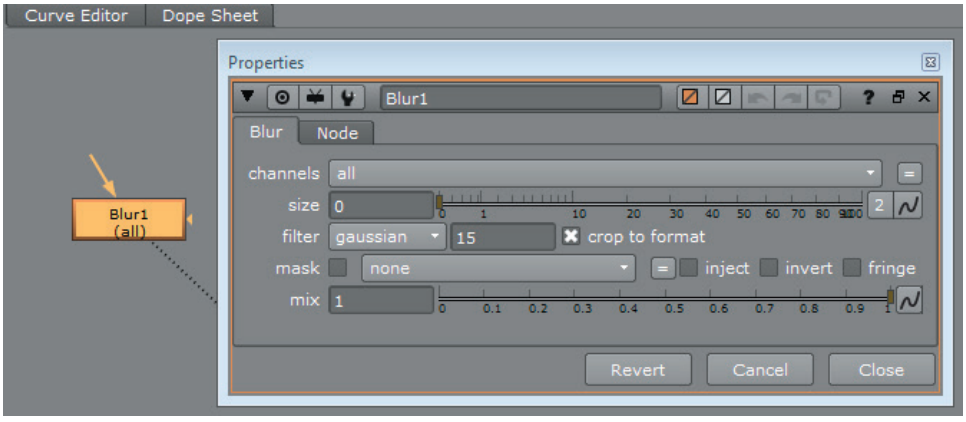


Figure 3-38 *Floating* ***Blur1*** *node properties panel*

You can set the number of properties panels that can be opened at a time in the **Properties Bin**. To do so, enter the number in the Max panels field located at the top left corner of the **Properties Bin**. To lock the **Properties Bin** and have all new properties panels appear in floating windows, choose the Lock Existing panels button from the **Properties Bin**. To empty the **Properties Bin**, choose the Remove all panels button from the **Properties Bin**, refer to Figure 3-39.

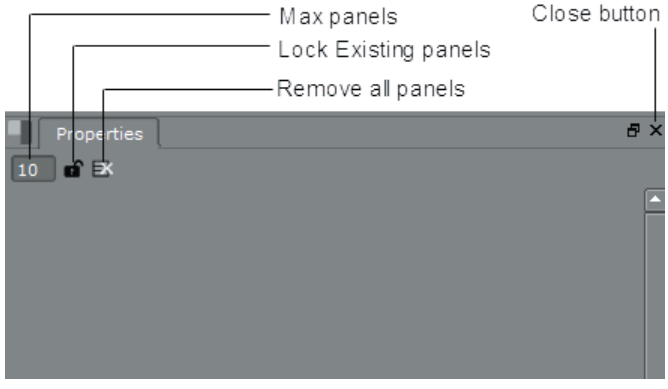


Figure 3-39 *The* ***Properties Bin***



Tip: *Press and hold ALT and then choose the Close (x) button of any of the panels to close all properties panels in the* ***Properties Bin****. Press and hold CTRL and then click on Close (x) button to close all properties panels except the one that is clicked.*

Standard Node Controls

There are some standard node controls that appear on all nodes. These node controls are discussed next:



This button is used to hide or show the tabs of the node. It also hides or shows the node's controls in the **Viewer#** panel.

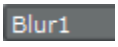


This button is used to place the node at the center of the **Node Graph** panel.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

 This button is used to center one of the node's inputs in the **Node Graph** panel.

 This button is used to create/load a saved node preset.



This field is used to enter a new name for the node.

 This button is used to change the color of the node tile in the **Node Graph** panel.

 This button is used to change the color used for the node's controls in the **Viewer#** panel.

 This button is used to undo the last change made to the node.

 This button is used to redo the last change made to the node.

 This button is used to revert all changes made after the properties panel was opened.

 This button is used to display a help tool tip for the selected node.

 This button is used to float a properties panel. Choose this button again to dock the properties panel back in the **Properties Bin**.

 This button is used to close the properties panel.

Three more buttons appear on a floating properties panel, refer to Figure 3-38. These buttons are discussed next:



This button is used to revert any changes made after the properties panel was opened.



This button is used to revert any changes made after the properties panel was opened and it also closes the properties panel. If you choose this button right after a node is created, the node is deleted from the **Node Graph** panel.



This button is used to close the properties panel.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Node Tab

This tab is found on the properties panel of every node, refer to Figure 3-40. The parameters in this tab are discussed next.

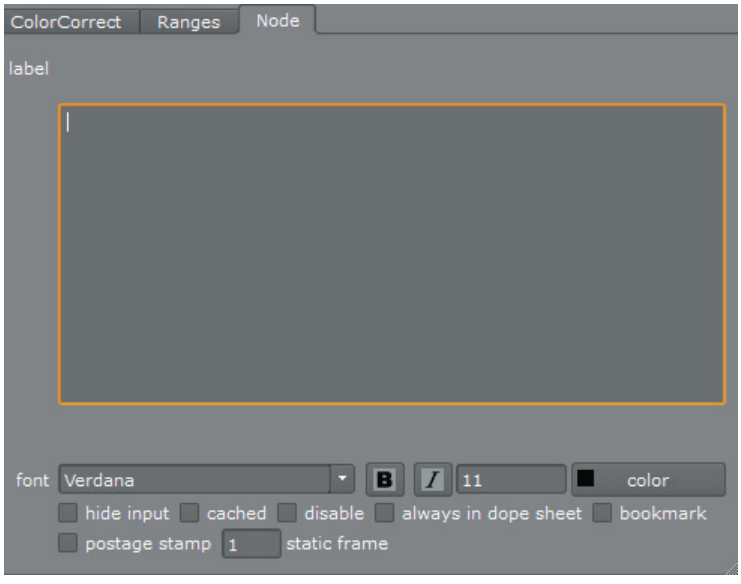


Figure 3-40 *Parameters in the* ***Node*** *tab*

label

This field is used to add comments about the node. You can use basic HTML markup in this field.

font

The options in this drop-down are used to change the font of the text displayed on the node. The bold and italic buttons are used to bold or italicize text displayed on the node.

note_font_size

This field is used to change the size of the text displayed on the node.

color

This button is used to change the color of text displayed on the node in the **Node Graph** panel.

hide input

This check box is used to hide the incoming pipe of the node.

cached

This check box is used to keep the output data of the node in the memory so that it can be read quickly.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

disable
This check box is used to disable or activate the node.

always in dope sheet
By default, the node will be displayed in the **Dope Sheet** panel when its properties panel is open. On selecting this check box, the node always opens in the **Dope Sheet** panel.

bookmark
When this check box is selected, the node is shown in the bookmark list.

postage stamp
This check box is used to display a thumbnail render of node's output on its tile in the **Node Graph** panel.

static frame
This field is used to specify the frame number to be used for generating the postage stamp.

Working with Properties Panel Controls
NukeX gives you various types of controls to edit values in the properties panel, refer to Figure 3-41. These controls are discussed next.

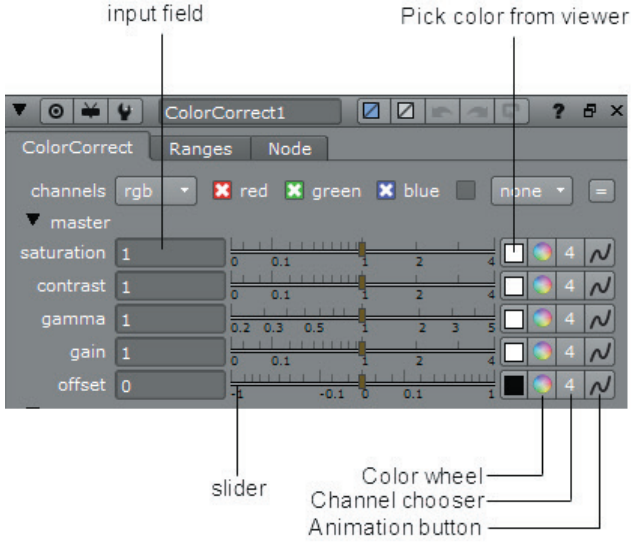


Figure 3-41 *The parameter controls in the* ***ColorCorrect1*** *node properties panel*

Fields
You can enter values directly in the input fields or use arrow keys (UP and DOWN) to increase or decrease the values. When arrow keys are used, the change in the value of the field depends on the initial position of the cursor. For example, if the current value of the field is 125.25

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

and you need to change it to 126.25; place your cursor after 2 and press UP ARROW once. Now, if you press DOWN ARROW twice, new field value will be 124.25. You can also enter expressions into the fields. Additionally, you can enter formulae in the input fields for quick calculations. For example, if you enter 1280/2 in a field and then press ENTER, NukeX will return a value of 640.

Tip: *You can also increase or decrease field values using the mouse wheel. To do so, place the cursor in the field and then scroll mouse wheel. If the mouse wheel is not available then press and hold ALT and drag the field values.*

Sliders

To set a value with the slider, drag the slider knob as required or click on the scale located beneath the slider. To reset a slider to its default value, press and hold CTRL and click on the slider.

Channels

By default, NukeX uses single value for the R, G, B, and A channels. For example, if in the **ColorCorrect1** node properties panel you set the **gain** field value to 1.2, this value will simultaneously affect the R, G, B and A channels, refer to Figure 3-42. You can choose the field's Channel chooser button to switch between single and multiple values. As a result, an array of input fields will be displayed for editing channel values individually. The number on the button indicates the number of input fields available, refer to Figure 3-43.

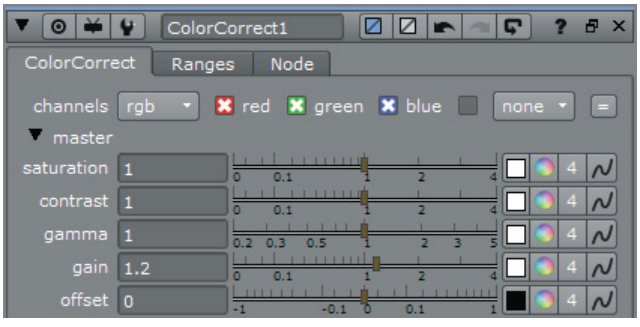


Figure 3-42 *The value **1.2** applied to all channels in the **gain** field of the **ColorCorrect1** properties panel*

Pick color from the viewer

The Pick color from the viewer button is used to input the color values. To pick the color from the **Viewer#** panel, choose the Pick color from viewer button; an eyedropper will be activated. Next, CTRL+click on the **Viewer#** panel to sample a color. To sample a color from the node's input while viewing its output, press and hold CTRL+ALT and click on the **Viewer#** panel. Press and hold SHIFT+LMB and drag the cursor on the **Viewer#** panel to sample a region instead of a point. As a result, a red rectangle will be drawn on the **Viewer#** panel and the sample color will be the average of the pixels inside that rectangle. Choose the button again to lock the selected color to prevent unwanted color sampling.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

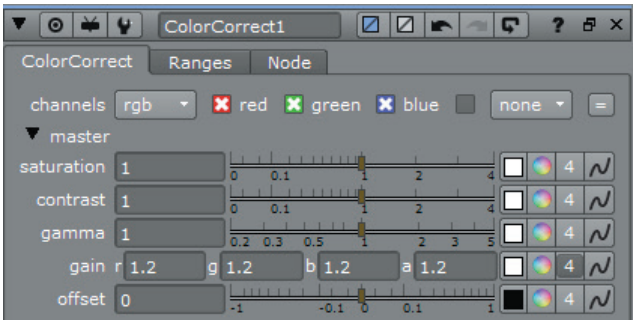


Figure 3-43 *Separate **r**, **g**, **b**, and **a** channels displayed for the **gain** field of the **ColorCorrect1** properties panel*

Color wheel

On choosing the field's Color wheel button, the color picker window will be displayed along with the color sliders and color wheel. The color sliders and color wheel in this window are used to set the precise color values. To make sliders horizontal in this window, resize the dialog box. When the size of the window is wide enough, sliders are placed horizontally, refer to Figure 3-44.

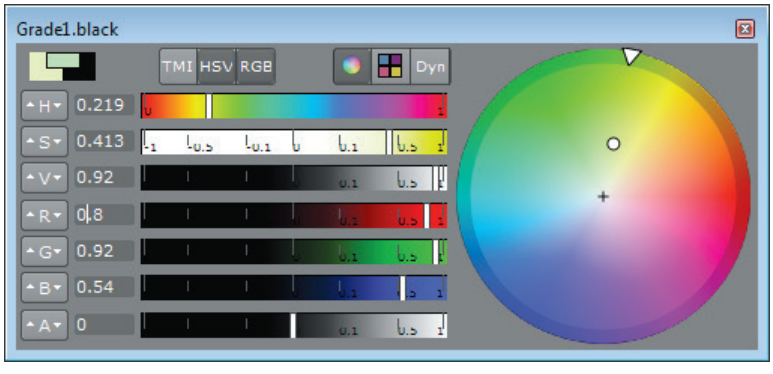


Figure 3-44 *The color picker window displayed on choosing the Color wheel button from the properties panel of the **Grade1** node*

To change the slider set displayed in the color picker window, choose **TMI** (Temperature, Magenta/Green, Intensity), **HSV** (Hue, Saturation, Value), or **RGB** (Red, Green, Blue) button from the color picker window, refer to Figure 3-44. To enable dynamic colors for the sliders background, choose the **Dyn** button. Choose the **Show the color wheel** button to cycle through color wheel states, the color wheel, and square, refer to Figure 3-45. Choose the color swatch button to show or hide the color swatches in the color picker window, refer to Figure 3-46.

Drag the white triangle located at the edge of the color wheel to adjust the hue, refer to Figure 3-47. Drag the white circle located inside the color wheel to adjust the saturation, refer to Figure 3-47.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

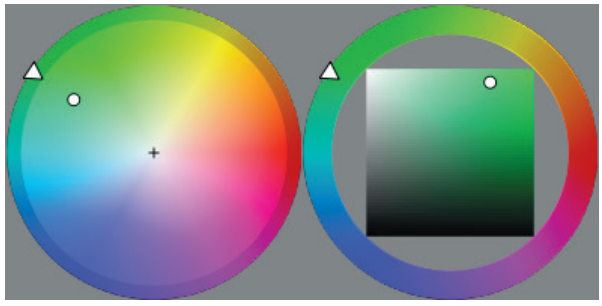


Figure 3-45 *The color wheel states*

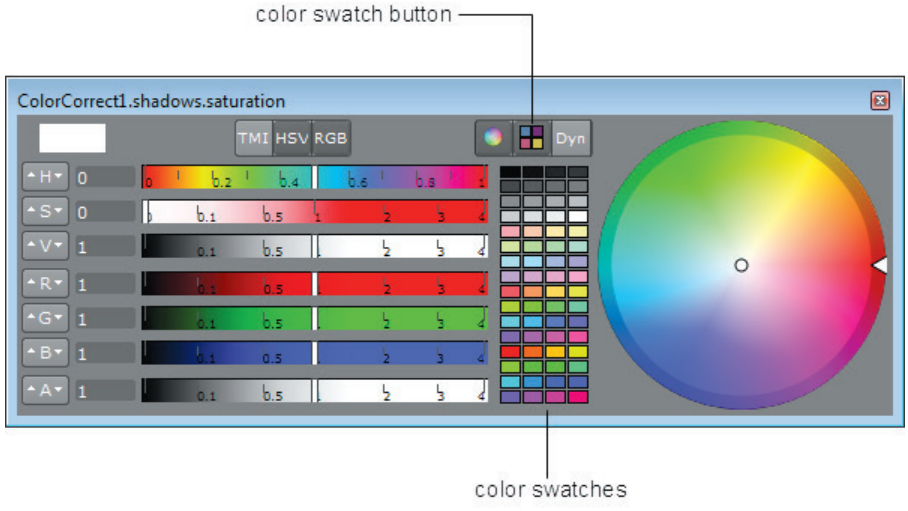


Figure 3-46 *Color swatches displayed in the color picker window on choosing the color swatch button*

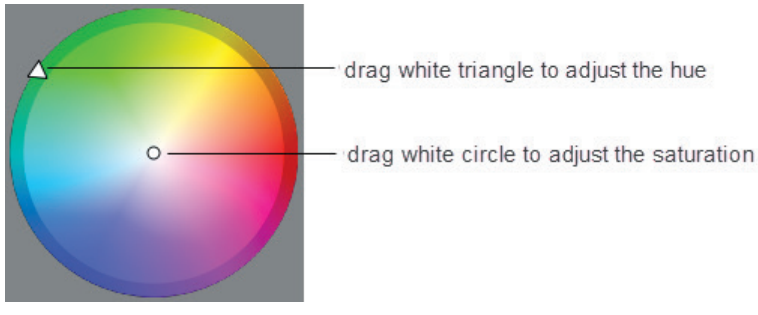


Figure 3-47 *Controls on the color wheel*

To affect only the hue of the selected color, press and hold CTRL and then drag the white triangle of the color wheel. Similarly, press and hold the SHIFT key and drag the white circle to change only the saturation, refer to Figures 3-48 and 3-49.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

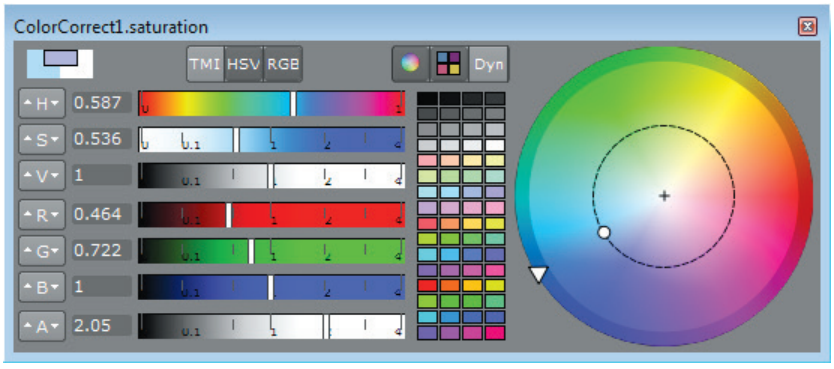


Figure 3-48 *Changing only the hue values*

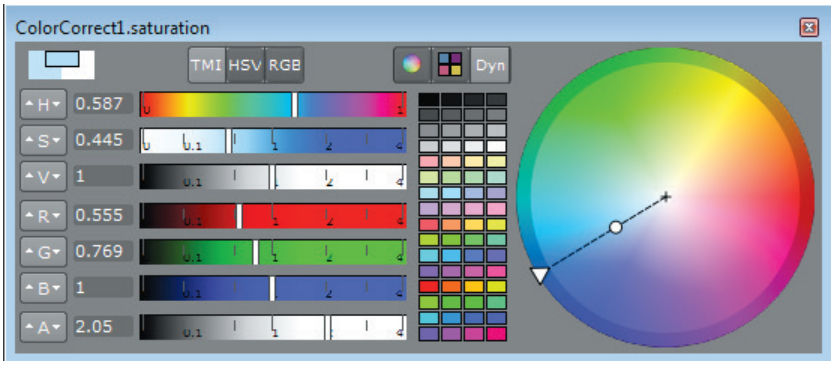


Figure 3-49 *Changing only the saturation values*

You can also zoom and pan inside the color wheel. To zoom in or out in the color wheel, press and hold ALT and then drag the cursor up or down using the middle mouse button. To pan in the color wheel, press and hold ALT and then drag the cursor on it. To reset the zoom or pan operation, press the middle mouse button on the color wheel. The slider labels on the color wheel window are used to increment color values precisely, refer to Figure 3-50. To increase the value by 0.01, right click on the label. To decrease the value by 0.01, left-click on the label. If you want to change the value by 0.1 or 0.001, use SHIFT or ALT. To open another color picker window while keeping the previously opened color picker window open, CTRL+click on the another parameter's color picker button.



Figure 3-50 *The slider labels*

WORKING WITH THE METADATA

Metadata is data within the data. It is information such as original bit depth, width, height, and so on of an image which is embedded in it. NukeX reads the embedded metadata information and passes it down to the process tree. You can change the metadata information in NukeX and also embed the new metadata to an image. You can access metadata nodes from the **Metadata** menu which is displayed on choosing the **MetaData** button from the **Nodes** toolbar. In this section, you will learn how to view, edit, and render the metadata.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Viewing Metadata

To view the metadata embedded to an image, select the node whose metadata you want to view and then choose the **MetaData** button from the **Nodes** toolbar; the **MetaData** menu will be displayed. Next, choose **ViewMetaData** from the menu; the **ViewMetaData#** node will be inserted in the **Node Graph** panel. You can view the metadata information in the **ViewMetaData** tab of the **ViewMetaData#** properties panel, as shown in Figure 3-51.

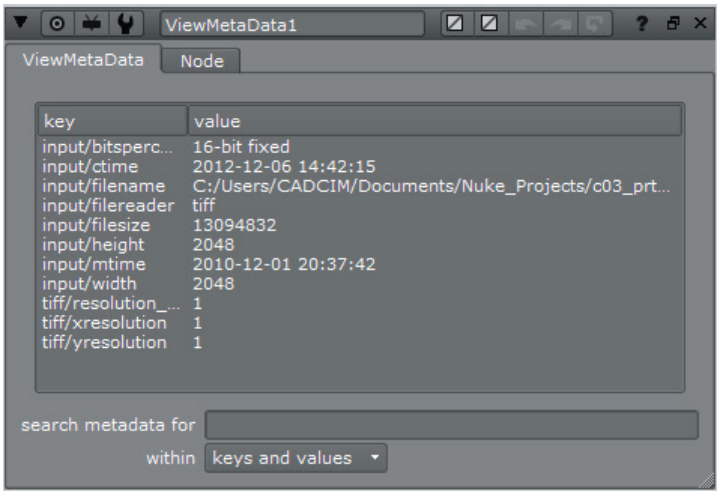


Figure 3-51 *The metadata information displayed in the* ***ViewMetaData1*** *node properties panel*

To filter the metadata list, you can use the **search metadata for** field in the **ViewMetaData#** node properties panel. To do so, enter a filter criteria in this field; the filtered information will be displayed in the **ViewMetaData** tab of the properties panel of the node. Fox example, if you enter **input** in the **search metadata for** field and then press ENTER; the keys and values having the word **input** will be displayed in the properties panel, refer to Figure 3-52. By default, filtering is done for both keys and values. However, you can limit it to keys or values by selecting the **keys only** or **values only** option from the **within** drop-down of the **ViewMetaData#** properties panel.



Tip: *The keys can be referenced while writing expressions.*



Note

1. While viewing the metadata information of a QuickTime file, the ***ViewMetaData*** *node does not display the gamma or bit depth, refer to Figure 3-53.*

2. If you add the ***ViewMetaData*** *node after the* ***Merge*** *node having two inputs* ***A*** *and* ***B****, you can specify the input whose metadata will flow downstream. To do so, select the* ***A*** *or* ***B*** *option in the* ***metadata from*** *drop-down in the* ***Merge*** *tab of the* ***Merge*** *node properties panel. You will learn about the* ***Merge*** *node later in this chapter.*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

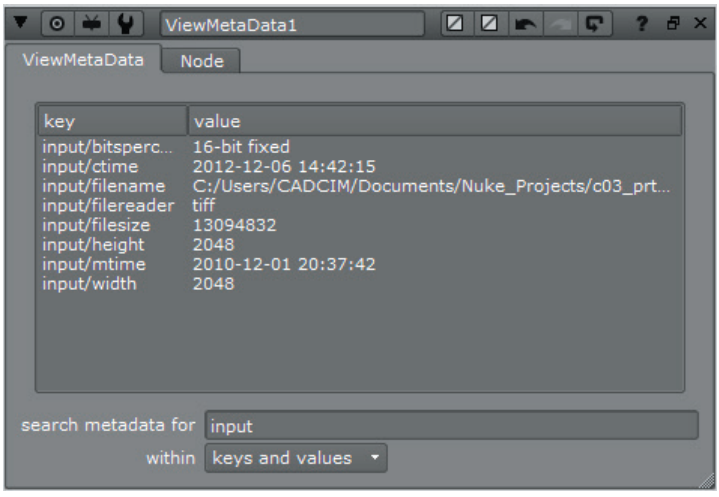


Figure 3-52 *The filtered metadata list displayed for key* ***input***

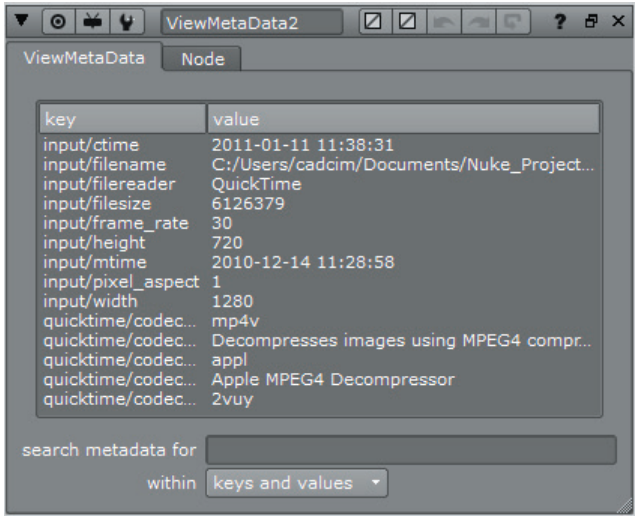


Figure 3-53 *The metadata of a QuickTime file displayed in the* ***ViewMetaData#*** *node properties panel*

Comparing Metadata

To compare metadata between the two inputs, add the **CompareMetaData** node from the **MetaData** menu and then connect the nodes that you need to compare with the **A** and **B** inputs of the **CompareMetaData#** node, refer to Figure 3-54. On doing so, the **CompareMetaData** node will display a list of keys where there is difference in the metadata of the two input images, refer to Figure 3-55.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

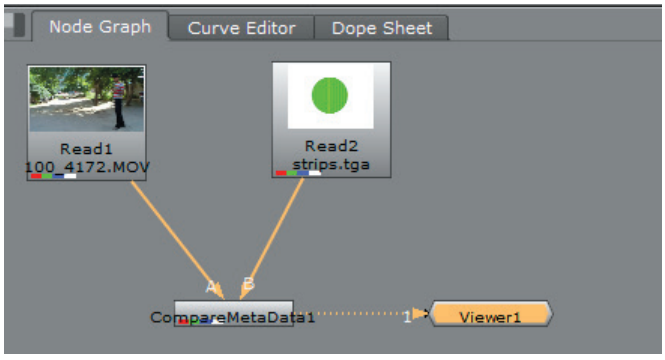


Figure 3-54 *The* ***Read1*** *and* ***Read2*** *nodes connected to the* ***CompareMetaData1*** *node*

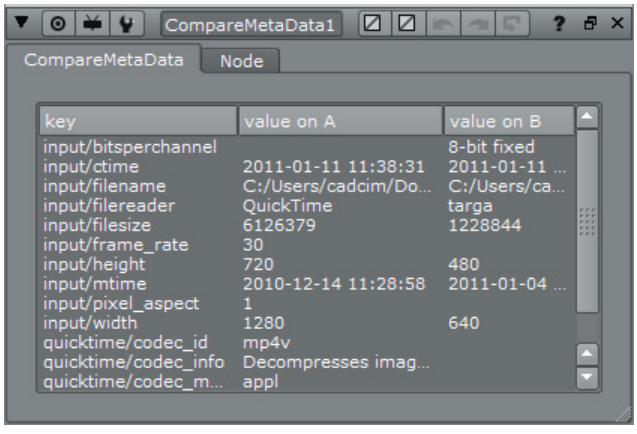


Figure 3-55 *The* ***CompareMetaData1*** *node displaying difference in the keys of the two input images*

Adding Metadata

To add metadata, select the node whose metadata you need to add and then choose the **ModifyMetaData** node from the **MetaData** menu; the **ModifyMetaData#** node will be connected with the selected node and its properties panel will be displayed. In this properties panel, choose the **+** button from the **ModifyMetaData** tab; a placeholder will be created in the metadata box, refer to Figure 3-56.

Next, double-click on the first cell under the **key** column header; the **Pick metadata key** dialog box will be displayed, refer to Figure 3-57. Enter a new name in the field located at the bottom of the dialog box and then choose the **OK** button. Now, double-click on the first cell under the **value** column header and then enter the value in it; the new key and its value will now flow downstream.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

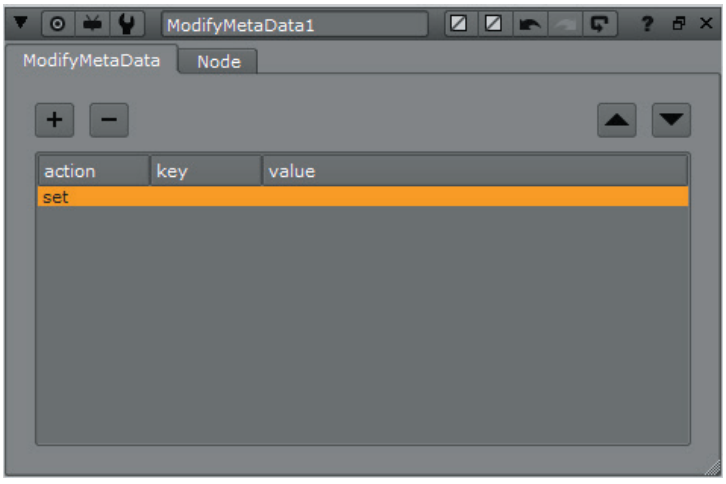


Figure 3-56 *The* ***ModifyMetaData1*** *node properties panel*

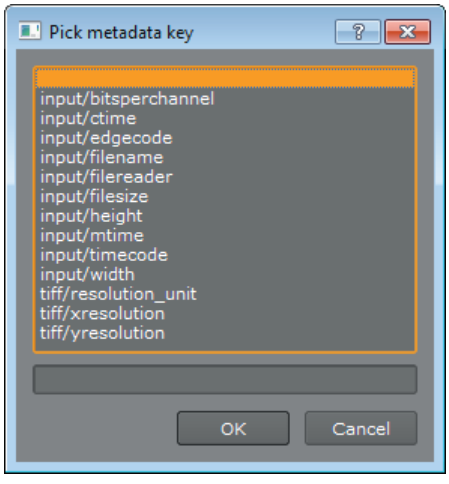


Figure 3-57 *The* ***Pick metadata key*** *dialog box*

Editing Metadata

To edit metadata, in the **ModifyMetaData#** node properties panel, choose the **+** button; a placeholder will be created in the **metadata** box, refer to Figure 3-56. Next, double-click on the first cell under the **key** column header; the **Pick metadata key** dialog box will be displayed, refer to Figure 3-57. In this dialog box, first select the key that you need to edit and then choose the **OK** button. Now, edit the value(s) and key(s) as required.

Removing Metadata

To remove metadata, choose the **+** button in the **ModifyMetaData#** node properties panel; a placeholder will be created in the **metadata** box. Next, double-click on the cell under **action** to display a flyout and then choose **remove** from the flyout. Invoke the **Pick Metadata key** dialog box, as discussed earlier. Select the keys to be removed from the dialog box and then choose

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

the **OK** button; the selected key will be removed from the metadata. To cancel an existing action choose the **-** (minus) button from the **ModifyMetaData** node properties panel.



Note

*When you delete a key from the metadata, it only affects the **ModifyMetaData** node calculations. It does not alter the embedded metadata in the input images.*

Copying MetaData

To copy metadata from one input to another, choose the **CopyMetaData** node from the **MetaData** menu; the **CopyMetaData#** node will be inserted in the **Node Graph** panel and its properties panel will be displayed with the **CopyMetaData** tab chosen by default, as shown in Figure 3-58.

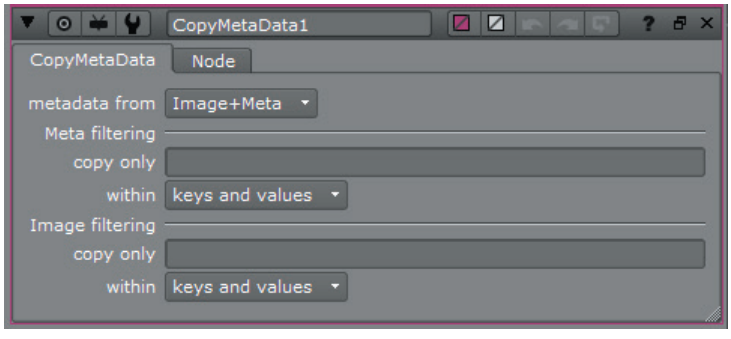


***Figure 3-58** The **CopyMetaData1** node properties panel*

The **CopyMetaData#** node has two inputs: **Image** and **Meta**. The **Image** input is used to connect to the image whose output you need to pass down. The **Meta** input is used to connect to the node whose metadata you need to copy to the output.

Once connections are made with a node in the **Node Graph** panel, choose the desired option from the **metadata from** drop-down of the **CopyMetaData#** node properties panel. By default, the **Image+Meta** option is selected in this drop-down. This option is used to add metadata from the **Meta** input to the metadata of the **Image** input. When the **Meta only** option is selected, NukeX only uses the metadata from the **Meta** input**.** The function of **Meta+Image** option is opposite to the **Image+Meta** option. The **Image only** option is used to take the metadata of the input image.

There are two areas in the **CopyMetaData** tab of the **CopyMetaData#** node properties panel: **Meta filtering** and **Image filtering**. The options in these areas are used to filter the metadata taken from the two inputs of the node. Fox example, if you enter **input** in the **copy only** field of the **Meta filtering** area and then press ENTER; the keys and values having the word **input** will be copied from the **Meta** input. By default, filtering is applied to both keys and values. However, you can limit it to only keys or values. To do so, select the desired option from the **within** drop-down. Similarly, you can filter the metadata from the **Image** input by setting the desired options in the **Image filtering** area.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Adding a Timecode to Metadata

To add a timecode to the metadata, choose the **AddTimeCode** node from the **MetaData** menu; the **AddTimeCode1** node will be inserted in the **Node Graph** panel and then its properties panel will be displayed with the **AddTimeCode** tab chosen in the **Properties Bin**, as shown in Figure 3-59.

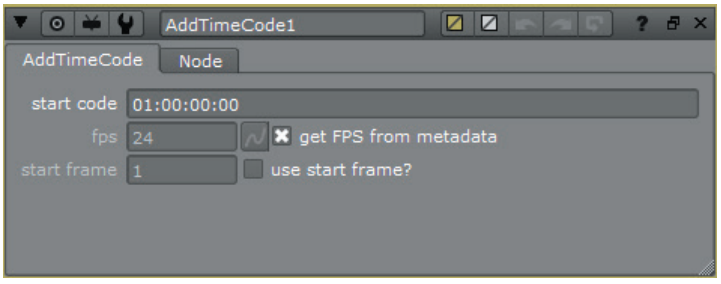


Figure 3-59 *The **AddTimeCode1** properties panel*

By default, timecode is set to 01:00:00:00 in the **start code** field of the properties panel, refer to Figure 3-59, which is timecode on the first frame. The value in the **fps** field is controlled by the **fps** value set in the **Project Settings** panel. If you need to manually set the **fps** value, clear the **get FPS from metadata** check box; the **fps** field will be enabled. Next, enter the desired value in the **fps** field. If you need to set different start frame than the first frame, select the **use start frame?** check box and then enter new value in the **start frame** field.

Rendering Metadata

To render the metadata, place the **Write** node at the desired place in the script and then set the required options in the **Write** node properties panel. NukeX only writes metadata information in the following formats: *EXR*, *CIN*, *JPEG*, and *DPX*. Select the format of file from the **file type** drop-down; the metadata option related to the file format will be displayed in the **Write** node properties panel. Next, set the option as required and then render the **Write** node.

COMPOSITING IMAGES

The compositing of images implies combining multiple images to create a single seamless image. The nodes in the merge category are used to composite two or more images. Most commonly used nodes in this category are discussed next.

Merge Node

The **Merge** node combines two input images based on the transparency (alpha channel) using various algorithms. The alpha channel is used to determine which pixels of the foreground image will be used for the composite. This node takes three inputs: **A**, **B**, and **mask**. The **A** input is used to connect foreground image to the **Merge** node. This image merges with the image that is connected to the **B** input. When you connect an image to the **A** input of the **Merge** node, an additional **A** input will be displayed on it, refer to Figure 3-60. Each input is named in the order it was connected with other nodes, **A1**, **A2**, **A3**, and so on. It means that you can connect as many images as you need on the **A** side of a **Merge** node. NukeX copies data from the **A** input to the **B** input. If you disconnect the node connected to the **A** input,

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

the data stream will still flow down as NukeX will use the **B** input. The **mask** input is used to connect a node to use as a mask.

The **Merge** node connects multiple images using various algorithms such as **multiply**, **overlay**, **screen**, and so on. To add a **Merge** node to the workspace, press the M key; the **Merge#** node will be inserted in the **Node Graph** panel and its properties panel will be displayed with the **Merge** tab chosen in the **Properties Bin**, refer to Figure 3-61. The options available in the **Merge** tab of the **Merge#** node properties panel are discussed next.

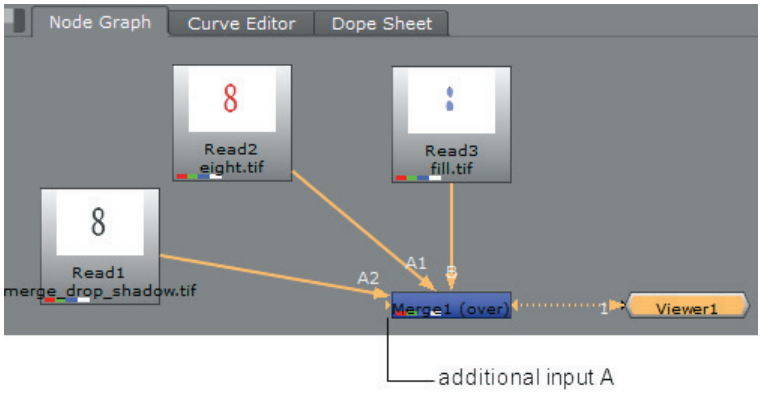


Figure 3-60 *Additional input **A** displayed on the **Merge1** node*

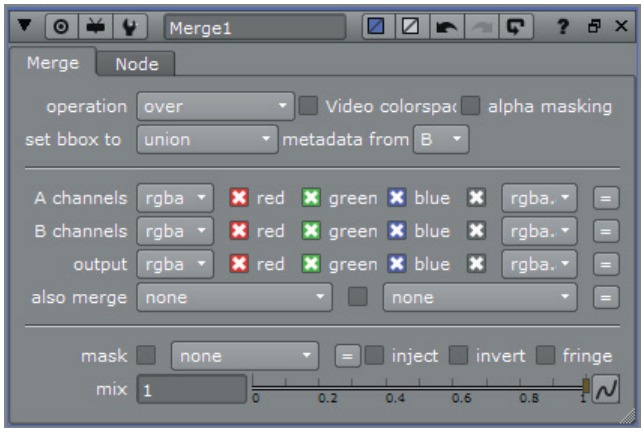


Figure 3-61 *The **Merge#** node properties panel*

operation

The options in the **operation** drop-down are used to set the algorithm to be used for merging the images. By default, the **over** algorithm is selected in this drop-down. It layers the image sequence connected to the **A** input over the image sequence connected to the **B** input according to the alpha channel present in the **A** input.

Tip: *To see the math formula for a particular merge algorithm, place the cursor over the **operation** drop-down; a tooltip will be displayed. This tooltip contains the information about the mathematics behind a merge operation.*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

When the **Video colorspace** check box located next to the **operation** drop-down is selected, NukeX converts all colors to the default 8-bit colorspace before compositing and then outputs them in linear colorspace. You can change the default colorspace for 8-bit files from the **Project Settings** panel. To do so, hover the mouse over the workspace and then press S; the **Project Settings** panel will be displayed. In this panel, choose the **LUT** tab; the **Default LUT settings** area will be displayed. Next, select the desired option from the **8-bit files** drop-down, refer to Figure 3-62.

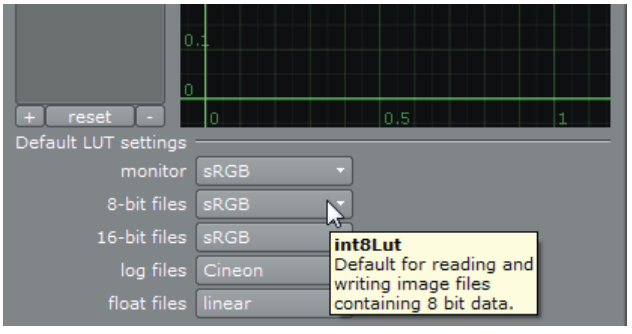


Figure 3-62 *Partial view of the **LUT** tab of the **Project Settings** panel*

On selecting the **alpha masking** check box located next to the **Video colorspace** check box, the image is processed according to the PDF/SVG spec. According to this spec, that the input image remains unchanged if the other composited image has zero alpha. The calculation applied to the alpha will be according to the following formula: *a+b-a*b*. If this check box is cleared, the formula applied to alpha will be same as that applies to other channels.



Note

*This check box will be disabled when it does not affect the operation selected from the **operation** drop-down or PDF/SVG.*

set bbox to

The options in this drop-down are used to set the bounding box. The bounding box defines the area of the frame that is having valid image data. It is used to speed up the processing. By default, full image area is the bounding box of the input image but if you crop a particular input, the bounding box will be reduced to the cropped area. The default option in this drop-down is **union**, the other three are: **intersection**, **A**, and **B**. These options are discussed next.

union

The **union** option combines the two bounding boxes from the **A** and **B** inputs. It resizes the output bounding box to fit the two input bounding boxes completely.

intersection

When you select the **intersection** option, the output bounding box will be the overlapping area of the two input bounding boxes.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

A
Select the **A** option to use the bounding box from the **A** input.

B
Select the **B** option to use the bounding box from the **B** input.

metadata from

The options in this drop-down are used to specify the node whose metadeta will flow down the process tree.

A channels

The options in the first **A channels** drop-down are used to specify which channels from the **A** input will be merged with the **B** input. The options in the second **A channels** drop-down are used to specify additional channel (alpha) to be merged with the **B** input. If you select **none** from the first **A channels** drop-down, the output of the **A** input will be black or zero. You can select check boxes on the right of the first **A channels** drop-down to select individual channels.

B channels

The options in the first **B channels** drop-down are used to specify which channels from the **B** input will be merged with the **A** input. The options in the second **B channels** drop-down are used to specify additional channel (alpha) to be merged with the **A** input. You can select the check boxes on the right of the first **B channels** drop-down to select individual channels.

output

The options in the first **output** drop-down are used to specify the output channels after the merge operation. The options in the second **output** drop-down are used to specify an additional output channel (alpha) after the merge operation. You can select the check boxes on the right of the first **output** drop-down to select individual channels.



Note

*There are four check boxes on the right of the **A channels**, **B channels**, and **output** drop-downs, namely **red**, **green**, **blue**, and Enable channel, refer to Figure 3-63. You can use these check boxes to keep or remove the channels from the merge calculations, as required. When the Enable channel check box is selected, the channels selected from the drop-down placed on the right of this check box are enabled. This check box is available in the properties panels of many nodes.*

also merge

The options in the first **also merge** drop-down are used to specify the channels that will be merged in addition to the channels specified from the **A channels** and **B channels** drop-downs. The options in the second **also merge** drop-down are used to specify the additional channel (alpha) to be merged. You can select the check boxes on the right of the first **also merge** drop-down to select individual channels. These check boxes appear when you select option other than **none** in the first **also merge** drop-down.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

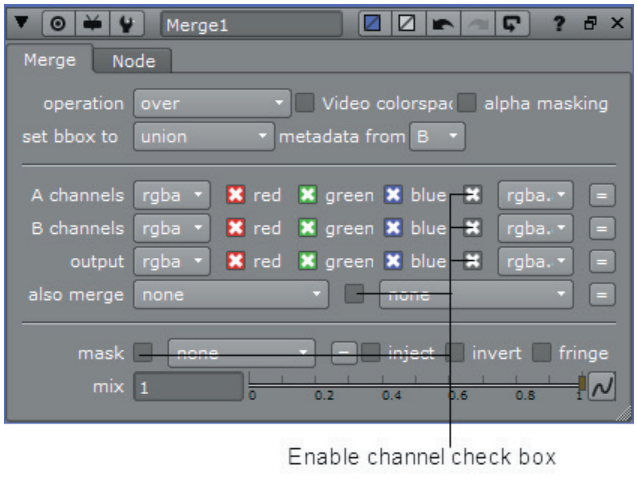


Figure 3-63 *The Enable channel check boxes displayed in the* ***Merge1*** *properties panel*

mask

The Enable channel check box located on the left of the **mask** drop-down is selected when you connect a mask to the **mask** input of the **Merge** node or select a channel from the **mask** drop-down. The options in this drop-down are used to select the channel that will be used as mask. When the **inject** check box is selected, NukeX copies the **mask** input to the predefined **mask.a** channel. The injected mask can be further used downstream in the process tree. By default, the merge is limited to the non-black areas of the mask. When you select the **invert** check box, the mask channel will be inverted and now merge will be limited to the non-white areas of the mask. The **fringe** check box is used to blur the edges of the mask.

mix

The **mix** parameter is used to blend the two merged inputs. When the value of this parameter is set to 0, only the **B** input will be displayed in the **Viewer#** panel. The full merge will be displayed when the value for this parameter is set to 1 which is the default value.

Premult Node

The **Permult** node is used to premultiply the input image. This node multiplies the rgb channels of the input image with its alpha channel. The alpha channel is used to determine which pixels of the foreground input image will be visible in the final composite. An input image that is not premultiplied is referred to as straight or unpremultiplied. If the black areas in the alpha channel are not black in the color channels, then the image is considered as Straight. Generally, most 3D rendered images are premultiplied. The **Merge** node expects premultiplied images so you should use the **Premult** node before any merge operation if input image is not premultiplied. This helps in removing artifacts such as fringes around a masked object. While color-correcting a premultiplied image, you should first connect an **Unpremult** node to the image and then perform color-correction. Next, connect a **Premult** node to get back to original premultiplied state for the merge operations. To add a **Premult** node to the

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Node Graph panel, select **Premult** from the **Merge** menu of the **Nodes** toolbar; the **Premult#** node will be inserted in the **Node Graph** panel and its properties panel will be displayed with the **Premult** tab chosen in the **Properties Bin**, refer to Figure 3-64. The options available in the **Premult** tab of the **Premult#** node properties panel are discussed next.

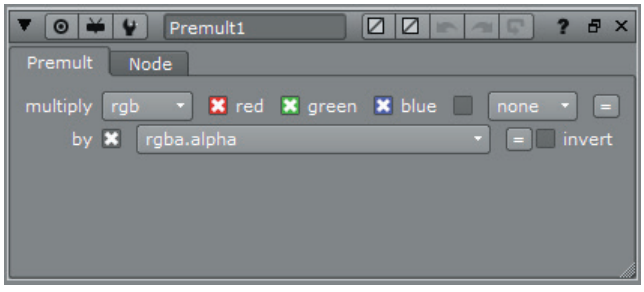


Figure 3-64 *The* ***Premult1*** *node properties panel*

multiply

The options in the first **multiply** drop-down are used to set the channels (generally rgb) to be multiplied with the alpha channel. To select the individual channels, you can select the check boxes available on the right of the **multiply** drop-down. The options in the second **multiply** drop-down are used to set the additional channel to be multiplied with the alpha channel.

by

If you select the Enable channel check box located on the left of the **by** drop-down, the channel set in it (generally alpha) is multiplied with the channels set using the **multiply** drop-downs. The **invert** check box is used to invert the output of the alpha channel.

Unpremult Node

The **Unpremult** node is used to divide the rgb channels of the input image by its alpha. To add a **Unpremult** node to the **Node Graph** panel, select **Unpremult** from the **Merge** menu; the **Unpremult#** node will be inserted in the **Node Graph** panel and its properties panel will be displayed with the **Unpremult** tab in the **Properties Bin**, refer to Figure 3-65. The options available in the **Unpremult** tab of the **Unpremult#** node properties panel are discussed next.

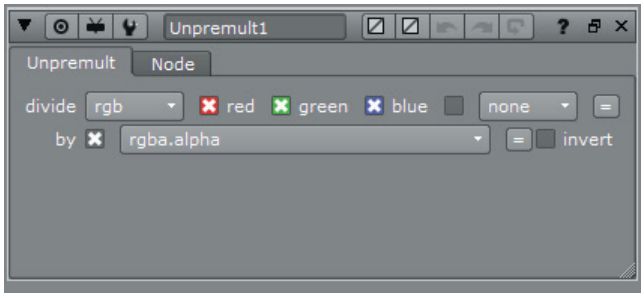


Figure 3-65 *The* ***Unpremult1*** *node properties panel*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

divide

The options in the first **divide** drop-down are used to set the channels (generally rgb) to be divided with the alpha channel. To select the individual channels, you can select the check boxes available on the right of the **divide** drop-down. The options in the second **divide** drop-down are used to set an additional channel to be divided with the alpha channel. The function of the **by** and **invert** check boxes is same as discussed in the **Premult** node.

WORKING WITH IMAGE NODES

Before you start compositing, you need to import image or image sequence to the **Node Graph** panel using the **Read** node. You can also generate images using NukeX nodes such as the **Constant** node. Once you load an image in the **Node Graph** panel, you can view the output of the **Read** node using the **Viewer** node. The **Write** node is used to write the output of all upstream nodes to the disk. All these nodes can be added using the **Image** menu which is displayed on choosing the **Image** button from the **Nodes** toolbar. The commonly used nodes in the **Image** menu are discussed next.

CheckerBoard

This node is used to generate a checkerboard image that can be used in place of a texture or background, refer to Figure 3-66. To add a **CheckerBoard** node to the **Node Graph** panel, choose **CheckerBoard** from the **Image** menu; the **CheckerBoard#** node will be inserted in the **Node Graph** panel and its properties panel will be displayed with **CheckerBoard** tab chosen in the **Properties Bin**, refer to Figure 3-67. The options available in the **CheckerBoard** tab of the **CheckerBoard#** node properties panel are discussed next.

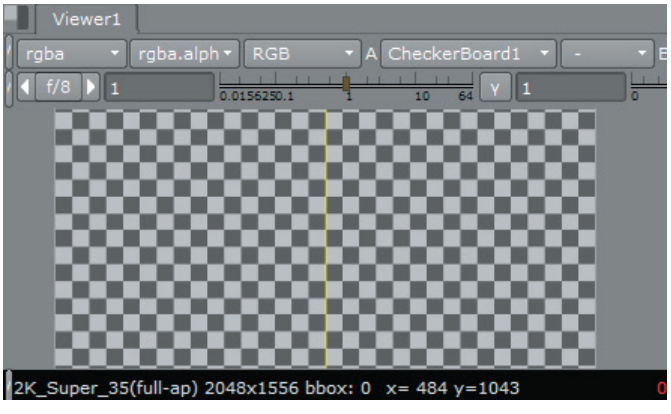


Figure 3-66 *The checkerboard pattern displayed in the* ***Viewer1*** *panel*

format

The options in this drop-down are used to set the format of the image that the **CheckerBoard** node generates. This setting is dependent on the format set in the **Project Settings** panel.

size

This parameter is used to set the size of the checkerboard boxes in pixels.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com



Note

On enabling the proxy mode, the size of the boxes is rounded to the nearest pixel. Therefore, the output of the ***CheckerBoard*** *node may look different than its full resolution output.*

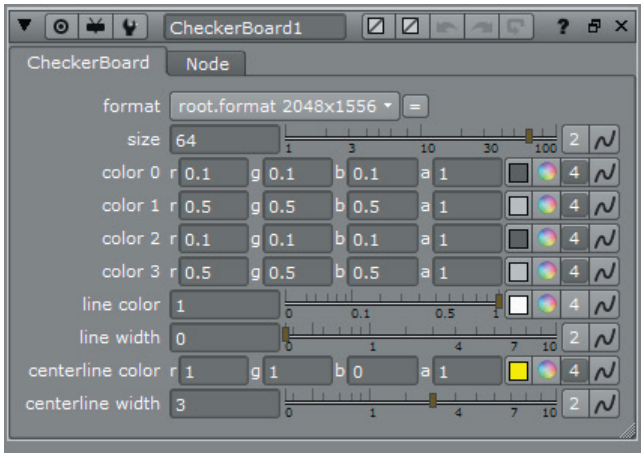


Figure 3-67 *The* ***CheckerBoard1*** *properties panel*

color 0

The **color 0** parameter is used to set the color of the boxes from the bottom left corner. The color specified using this parameter starts at (0,0) and then affects every alternate box and row. Similarly, **color 1**, **color 2**, and **color 3** start at (1,0), (1,1), and (0,1), respectively and affect every alternate box and row. Figure 3-68 shows output of the **CheckerBoard1** node with the **color 0**, **color 1**, **color 2**, and **color 3** parameters set to red, green, blue, and yellow colors, respectively.

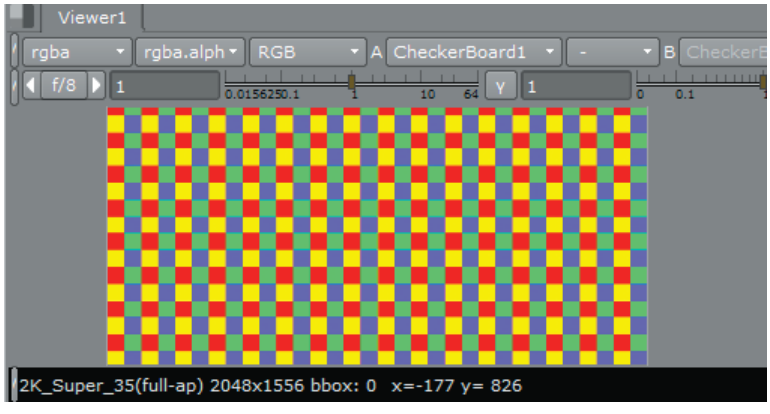


Figure 3-68 *The output of the* ***CheckerBoard1*** *node*

line color

This parameter is used to set the color of the outline of the boxes.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

line width

This parameter is used to set the width of the outline of the boxes in pixels. Figure 3-69 shows the output of the **CheckerBoard1** node with **line color** set to red and **line width** to 10.

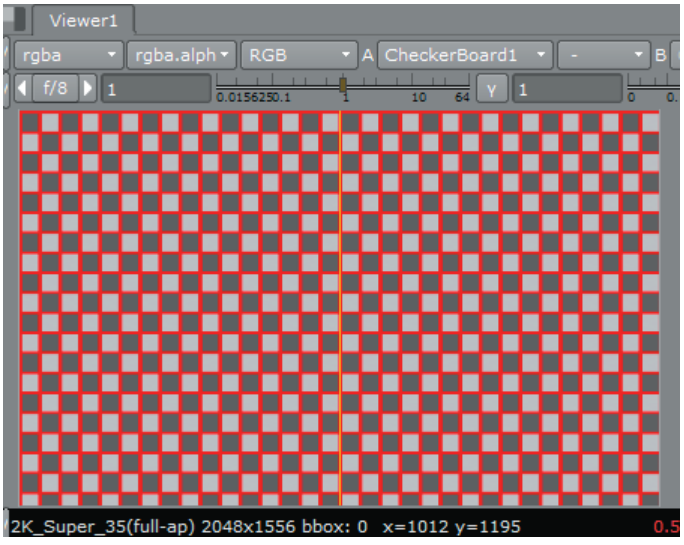


Figure 3-69 *The output of the* ***CheckerBoard*** *node displayed on changing the values of the* ***line color*** *and* ***line width*** *parameters*

centerline color

This parameter is used to set the color of center lines on the checkerboard.

centerline width

This parameter is used to set the width of the center lines in pixels. Figure 3-70 shows the output of the **CheckerBoard1** node on setting the **centerline width** parameter to 10.

Constant Node

This node is used to generate a constant color image. To add the **Constant** node to the **Node Graph** panel, choose **Constant** from the **Image** menu; the **Constant#** node will be inserted in the **Node Graph** panel and its properties panel will be displayed with the **Constant** tab chosen in the **Properties Bin**, refer to Figure 3-71. The options available in the **Constant** tab of the **Constant#** node properties panel are discussed next.

channels

The options in the first **channels** drop-down are used to select the channels that will be included in this node and will be affected by it. You can select individual channels by selecting the check boxes placed on the right of the drop-down. The option in the second **channels** drop-down corresponding are used to select an additional channel that will be affected by this node.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

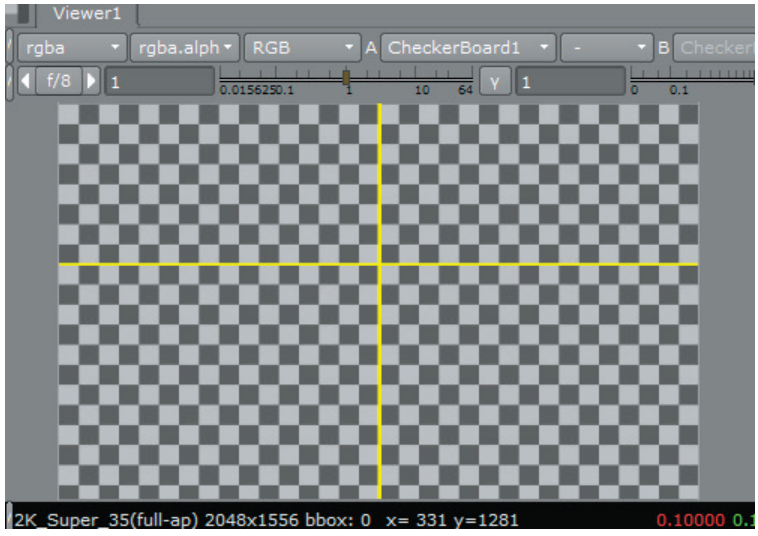


Figure 3-70 *The output of the* ***CheckerBoard*** *node displayed on changing the value of the* ***centerline width*** *parameter*

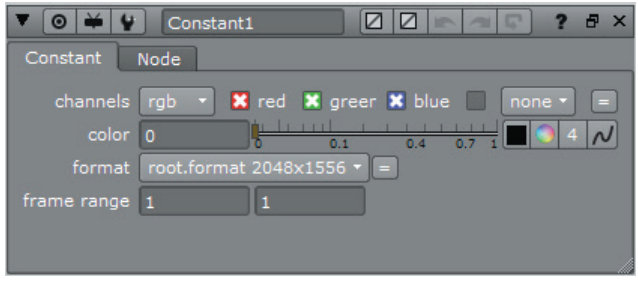


Figure 3-71 *The* ***Constant1*** *node properties panel*

color

This parameter is used to set the output color of the **Constant** node.

format

The options in this drop-down are used to set the format of the image that the **Constant** node generates. By default, the option selected in this drop-down is the format specified in the **Project Settings** panel.

frame range

The two fields corresponding to the **frame range** parameter are used to set the first and last frames of the frame range for animating the output of the **Constant** node.

Viewer Node

This node is used to view the output of connected process nodes in the **Viewer** panel. This node does not affect the data stream, it just displays the output. You can insert as many **Viewer**

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

nodes as you need in a script. To add the **Viewer** node to the **Node Graph** panel, choose **Viewer** from the **Image** menu or press CTRL+I. You can connect upto ten process nodes to a single **Viewer** node. To cycle through the views, put the cursor on the **Viewer#** panel. Next, press the UP or DOWN arrow to quickly cycle through the views. Alternatively, you can press the numbers (0, 1-9) corresponding to the port number of the **Viewer#** node. Figure 3-72 shows the **Blur1** node connected to the port **2** of the **Viewer1** node. Figure 3-73 shows the **Viewer1** node connected to the four process nodes.

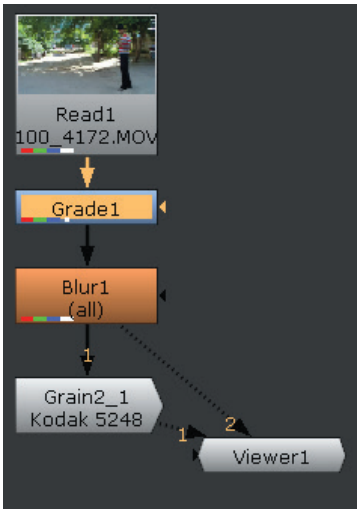


Figure 3-72 *The **Blur1** node connected to the port number **2** of the **Viewer1** node*

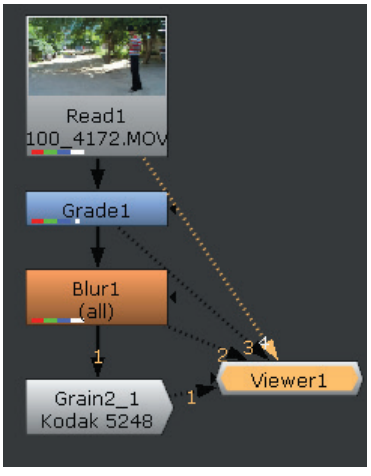


Figure 3-73 *All four nodes connected to the **Viewer1** node*

Read Node

This node is used to load images from the disk. This node converts all imported images to the native 32-bit linear RGB workspace. It supports file formats such as Cineon, TIFF, PSD, OpenEXR, HDRI, RAW Camera data, and so on. To add a **Read** node to the **Node Graph** panel, choose **Read** from the **Image** menu or press R; the **Read File(s)** dialog box will be displayed. In this dialog box, navigate to the desired file and then choose the **Open** button; the **Read#** node will be inserted in the **Node Graph** panel and its properties panel will be displayed with the **Read** tab chosen in the **Properties Bin**, as show in Figure 3-74. Various options available in the **Read#** node properties panel are discussed next.

Read Tab

The options available in this tab are used to read files from the disk, set format, set proxy and proxy format, set frame range, and colorspace. These options are discussed next.

file

This parameter displays the path of the file which is loaded using the **Read#** node. To change the file, you can click on the folder icon located next to this parameter; the **Read#:Replace** dialog box will be displayed. Next, navigate to the desired file and then choose the **Open** button to replace the file.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

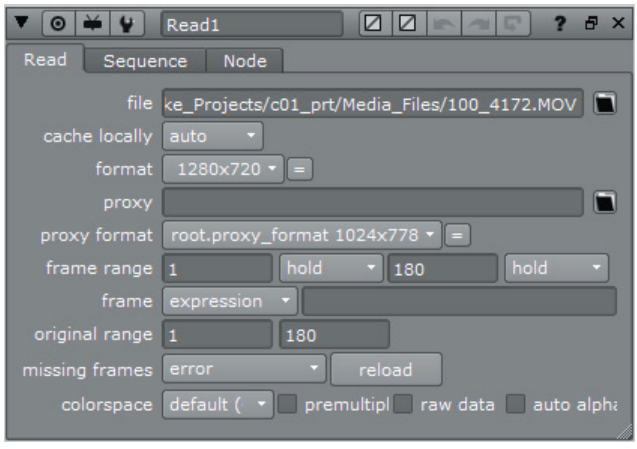


Figure 3-74 *The* ***Read1*** *node properties panel*

cache locally

The options in this drop-down are used to set the option for the local caching in a specified folder. Local caching helps in faster reloading of the files. To set the location of the folder for local cache, choose **Edit > Preferences** from the menu bar; the **Preferences** dialog box will be displayed. In this dialog box, you can set the local cache folder by modifying the **localise to** parameter value. Next, choose the **Save Prefs** button to save the changes made and then close the dialog box. The **cache locally** drop-down has three options, namely **auto**, **always**, and **never**.

format

The options in this drop-down are used to set the size and pixel aspect ratio of the loaded image. By default, NukeX matches it with the size and pixel aspect information stored in the header of the loaded image.

proxy

This parameter is used to set path for the proxy file. Generally, a proxy file is a low res version of the full res file and is used when proxy mode is on and the required resolution is less than or equal to the size of the file.

proxy format

The options in the **proxy format** drop-down are used to select the size and aspect ratio of the proxy file.

frame range

The fields corresponding to the **frame range** parameter are used to set the first and last frames of the image sequence. There are two drop-downs corresponding to the **frame range** parameter. These drop-downs are used to set the behavior of the **Read#** node while calculating the frames outside the range specified for the fields. By default, **hold** is selected in these two drop-downs. As a result, NukeX holds the first and last frames for out of range frames specified by the **first** and **last** fields. Other three options available are **loop**, **bounce**, and **black**. The **loop** option is used to continuously loop the sequence

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

for out of range frames. The **bounce** option is used to loop the sequence repeatedly back and forth for out of range frames. The **black** option is used to replace out of range frames with black frames.

frame

The options in the **frame** drop-down are used to set the frame mode. By default, the **expression** option is selected in this drop-down. You can enter an expression in the field located next to this drop-down. If the **start at** option is selected, the playback will not start until the playhead reaches the frame specified in the **frame** field. If you select the **offset** option, the frame displayed will be offset by the value specified in the field located next to the **frame** drop-down.

original range

The fields corresponding to the **original range** parameter are used to set the original frame range of the loaded images.

missing frames

The options in the **on_error** drop-down corresponding to the **missing frame** parameter are used to specify the action needed to be taken if the **Read** node encounters any error in loading the frames. By default, the **error** option is selected in the **missing frame** drop-down. As a result, NukeX displays **No Such file or directory** error message in the **Viewer** panel. If the **black** option is selected from the drop-down, the missing frame is filled with the black color. When the **Checkerboard** option is selected from this drop-down, the missing frame is filled with a checkerboard pattern. The **nearest frame** option from this drop-down is used to replace the missing frame by the nearest frame in the sequence. The **reload** button is used to re-read images from the disk.

colorspace

The options in the **colorspace** drop-down are used to specify the lookup table (LUT) that is used to convert values that NukeX uses internally. By default, the **default** option is selected in this drop-down. As a result, NukeX collects information from the header of the image. When the **premultiplied** check box is selected and alpha channel is available in the input image, NukeX first divides the color channels with the alpha channel before converting the colorspace and then multiply it with the alpha channel. This helps in removing artifacts from the input images. If you select the **raw data** check box, NukeX does not convert the data.

Sequence Tab

The options in this tab are used to import a sequence script. These options are discussed next.

import sequence

This button is used to import a sequence script. This script builds a frame sequence list from an EDL file. On choosing this button, the **Sequence File** dialog box will be displayed.

Next, you need to navigate to the script and then choose the **Open** button to import the script.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

frame sequence
This field is used to access multiple sequences using the single read node where the naming of sequences does not follow the numeric sequence.

timecode
This parameter is used to display the timecode if it is included in the image. It displays timecode of the last opened file.

edge code
This parameter is used to display the edge code if it is in the image. It displays the edge code of the most recently opened image.

RENDERING

Rendering is a process in which the result of all the upstream nodes is calculated and then saved to the disk. Generally, the **Write** node is placed at the end of the process tree to render the final output of the composition. However, you can place it anywhere in the tree as it has both the input and output connectors. The uses of the **Write** node and naming convention of the rendered sequence are discussed next.

Write Node

The **Write** node supports multiple file formats such as Cineon, OpenEXR, TIFF, and so on. To add a **Write** node to the **Node Graph** panel, select the last node in the process tree and then choose **Write** from the **Image** menu; the **Write#** node will be connected with the selected node in the **Node Graph** panel and its properties panel will be displayed with the **Write** tab chosen in the **Properties Bin**, refer to Figure 3-75. The options available in the **Write#** node properties panel are discussed next.

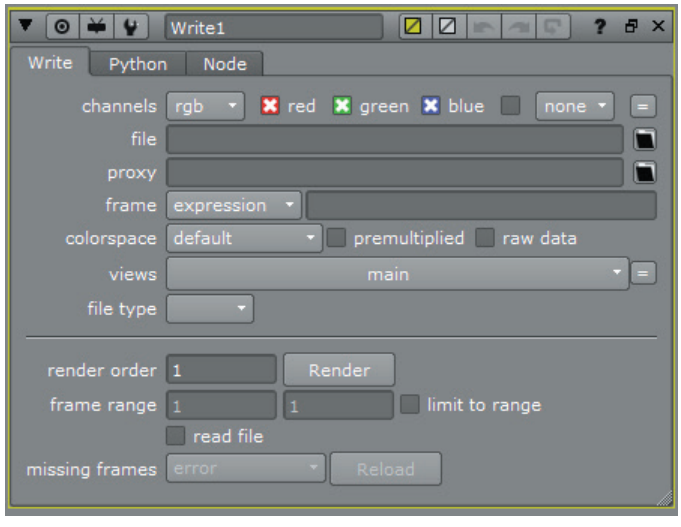


*Figure 3-75 The **Write1** node properties panel*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Write Tab

The options in this tab are used to specify the render settings. These options are discussed next.

channels

The options in the first **channels** drop-down are used to select the channels that the **Write** node will write. You can include individual channels in the output by selecting the check boxes placed at the right of the first **channels** drop-down. The options in the second **channels** drop-down are used to select an additional channel (alpha) that the **Write** node will write.

file

This parameter is used to define the path and name of the file. To do so, click on the folder icon corresponding to this parameter; the **Write1: Select file(s)** dialog box will be displayed. In this dialog box, navigate to the folder where you want to save the file and then enter the name of the file in the pathname field. Next, choose the **Save** button to close the dialog box.

proxy

The **Write** node executes all renders at full or proxy resolution depending on the currently active state. If proxy mode is enabled then you can toggle between resolutions using the CTRL+P keys. The **proxy** parameter is used to specify the path and name of the proxy file.

frame

The function of the **frame** parameter is same as that of the **frame** parameter of the **Read** node.

colorspace

The function of the **colorspace** parameter is same as that of the **colorspace** parameter of the **Read** node.

views

The options in this drop-down are used to select the view to be rendered.

file type

The options in this drop-down are used to set the type of file to be rendered. On selecting any option from this drop-down, more file specific parameters appear on the **Write** node properties panel. Figure 3-76 shows additional parameters in the **Write** node properties panel on selecting **tiff** from the **file type** drop-down.

render order

When multiple **Write** nodes are available in the script and are rendered together, then their rendering order is determined by the value specified in the field corresponding to this parameter. Choose the **Render** button to start the rendering process.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

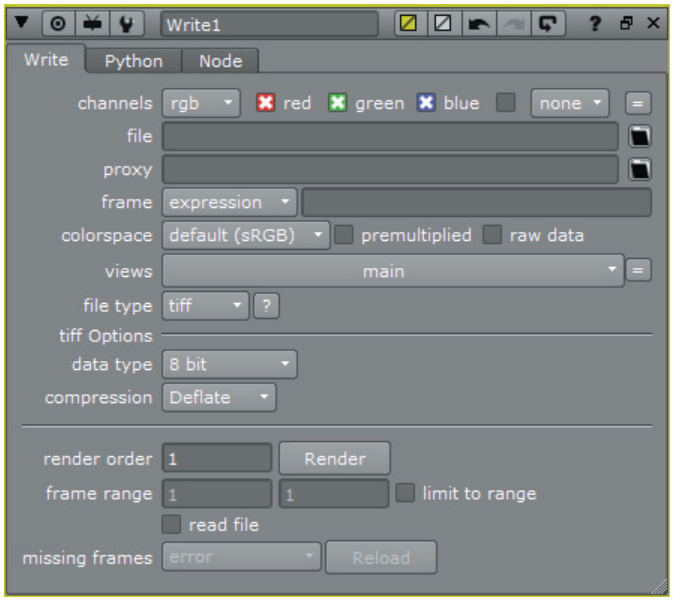


Figure 3-76 *Additional parameters displayed in the* ***Write1*** *node properties panel*

frame range

By default, the fields corresponding to the **frame range** parameter are disabled. Select the **limit to range** check box to enable them. When this check box is enabled, the **Write** node will only execute frames within the range specified in the fields. For frames outside the specified range, the **Write** node will act as if it is disabled.

read file

By default, the **read file** check box is cleared. Select this check box; the **missing frames** parameter will be enabled. The function of this parameters is same as that of the **Read** node. On selecting this check box, the **Write** node ignores its input. The output of the node comes from the rendered file.

Python Tab

The options in this tab are used to automatically execute the Python functions when some events occur in a script. These options are discussed next.

before render

This parameter is used to run functions before rendering starts.

before each frame

This parameter is used to run functions prior to starting rendering of each individual frame.

after each frame

This parameter is used to run functions after rendering is done in each frame. These functions are not called if render aborts.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

after render
This parameter is used to run functions after rendering of all frames is done.

Rendered Sequences

In NukeX, you can render a file in two ways: by creating a single file such as *.MOV* or by rendering a sequence. When you render a sequence (recommended) such as tiff, one tiff image is created per frame and that image is identified by the frame number. For example, render file of the frame 31 will be something like *name_of_file.00031.tiff*. To get padding in the rendered sequence, you need to specify a padding structure. For frame numbers, you can either use # for each digit or use printf-style formatting such as %04d. Here, 04 means number of digits to be used. To get a naming conventions such as *name_of_file.00031.tiff* you can use *name_of_file.#####.tiff* or *name_of_file.%05d.tiff* padding structure.

TUTORIALS

Before you start the tutorials, you need to download the c*03_nuke_7_tut.zip* file from *www.cadcim.com*. The path of the file is as follows: *Textbooks > Animation and Visual Effects > NukeX > The Foundry NukeX 7 for Compositors.* Next, you need to extract the contents of the zip file. To do so, navigate to the *Documents* folder and extract the contents of the downloaded zip file to *\Documents\Nuke_Projects*.

Tutorial 1

In this tutorial, you will use a simple A on B algorithm to composite a footprint. Figures 3-77 through 3-82 show the *f1.png* to *f6.png* images. These images will be used to composite the footprint, as shown in Figure 3-83. **(Expected time: 40 min)**



Figure 3-77 *The f1.png image*



Figure 3-78 *The f2.png image*



Figure 3-79 *The f3.png image*



Figure 3-80 *The f4.png image*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com



Figure 3-81 *The f5.png image*



Figure 3-82 *The f6.png image*

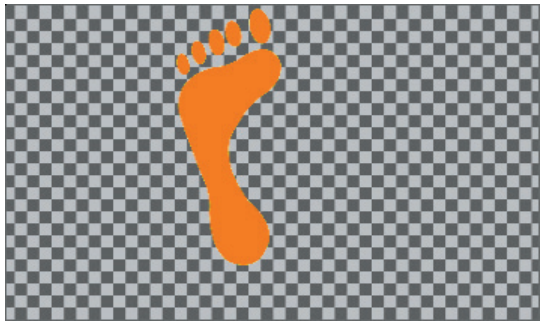


Figure 3-83 *The footprint composite*

The following steps are required to complete this tutorial:

a. Set the project.
b. Import the images.
c. Merge images with background.
d. Render and save the script.

Setting the Project

As the resolution of the image used in this tutorial is NTSC 720x480, you need to set the project accordingly.

1. Choose **File > New** or press CTRL+N; a new NukeX screen is displayed. Next, choose **File > Save** from the menu bar; the **Save script as** dialog box is displayed. In this dialog box, navigate to *\Documents\Nuke_Projects\c03_tut* and then append **c03_tut01** to the path displayed in the pathname field. Next, choose the **Save** button to save the script.

2. Hover the cursor over the **Node Graph** panel and press S; the **Project Settings** panel is displayed in the **Properties Bin**. Make sure the **Root** tab is chosen in it. Next, select **new** from the **full size format** drop-down, refer to Figure 3-84; the **New format** dialog box is displayed, as shown in Figure 3-85. In this dialog box, set the values of the parameters as given next:

   name: **NTSC720x480** file size w: **720** file size h: **480**
   pixel aspect: **0.91**

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

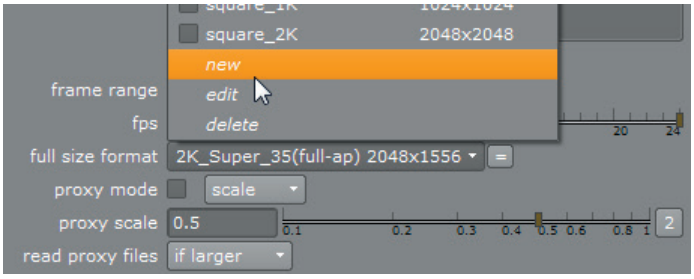


Figure 3-84 *Partial view of the* ***full size format*** *drop-down*

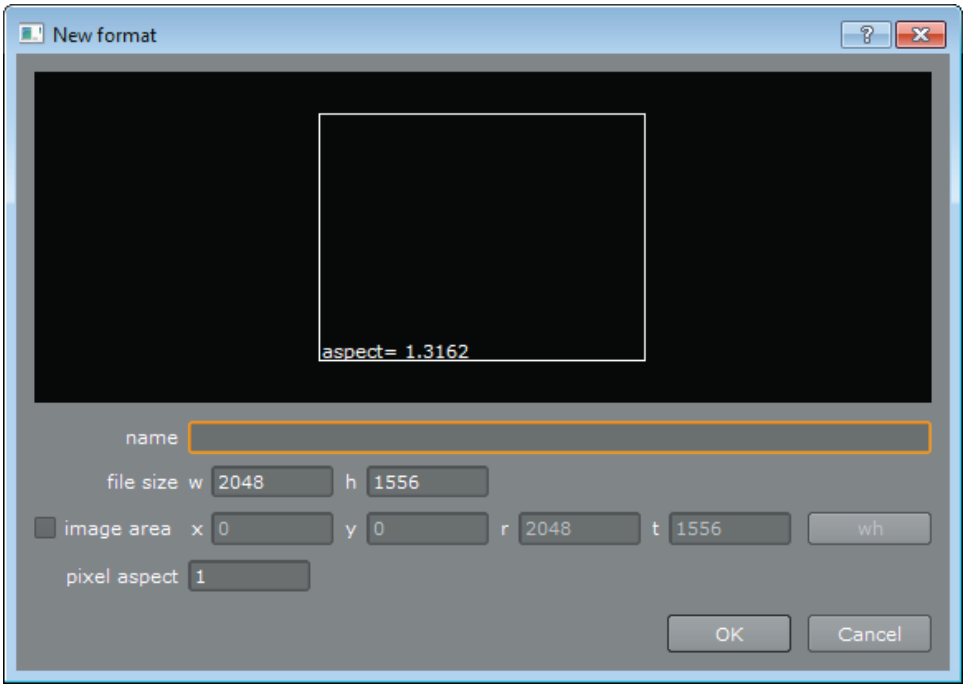


Figure 3-85 *The* ***New format*** *dialog box*

Next, choose the **OK** button to close the dialog box. You will notice that new format created is displayed in the **full size format** drop-down as well as in the **Viewer1** panel.

Importing the Images

In this section, you will import images from *f1.png* to *f6.png* and then bookmark the *Nuke_Projects* folder for quick access.

1. Choose the **Image** button from the **Nodes** toolbar; the **Image** menu is displayed. Next, choose **Read** from this menu; the **Read File(s)** dialog box is displayed. In this dialog box, choose **Home > Documents > Nuke_Projects**.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Next, you need to bookmark the *Nuke_Projects* directory. This will help to navigate to this directory with a single click when you import another image.

2. Right-click on the left pan of the **Read Files(s)** dialog box; a shortcut menu is displayed. Next, choose **Add** from the shortcut menu; the **Edit favorite** dialog box is displayed, as shown in Figure 3-86.

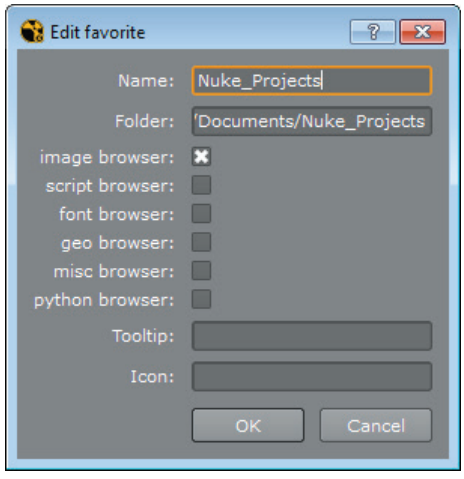


Figure 3-86 *The* ***Edit favorite*** *dialog box*

3. Change the name of the bookmark in the **Name** field or use the default name. Next, select the **script browser**, **geo browser**, **misc browser**, **font browser**, and **python browser** check boxes and then choose the **OK** button; a new entry *Nuke_Projects* is displayed in the directory list on the left of the **Read File(s)** dialog box.

4. Choose **c03_tut > Media_Files > footprint**. Notice that the **sequences** check box is selected by default at the lower left area of the dialog box. As a result, all six files are displayed as a single entry in the Directory List section of the dialog box. Clear the **sequences** check box; all six files are displayed as separate entries. Next, select **f1.png** and choose the **Open** button; the **Read1** node is inserted in the **Node Graph** panel. Also, the **Read1** node properties panel is displayed in the **Properties Bin**.

5. In the **Read** tab of the **Read1** node properties panel, select the **premultiplied** check box, refer to Figure 3-87.

6. Similarly, import all other five images (*f2.png* to *f6.png*) to bring them to the **Node Graph** panel. Figure 3-88 displays all the **Read** nodes in the **Node Graph** panel.

Merging Images with Background

In this section, you will merge the footstep images with the background. The background will be created using the **CheckerBoard** node.

1. Click on the empty area of the **Node Graph** panel to deselect the selected node if any. Next, choose the **Image** button from the **Nodes** toolbar; the **Image** menu is displayed.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Choose **CheckerBoard** from the menu; the **CheckerBoard1** node is inserted in the **Node Graph** panel and its properties panel is displayed in the **Properties Bin**, as shown in Figure 3-89.

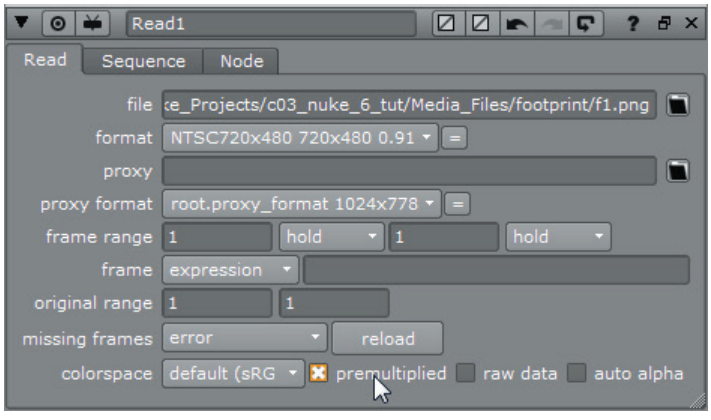


Figure 3-87 *The premultiplied check box selected in the* ***Read1*** *node properties panel*

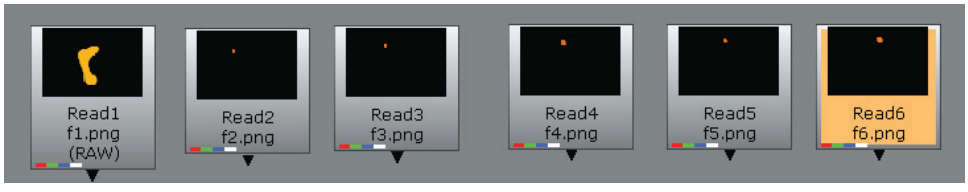


Figure 3-88 *Images imported in the* ***Node Graph*** *panel*

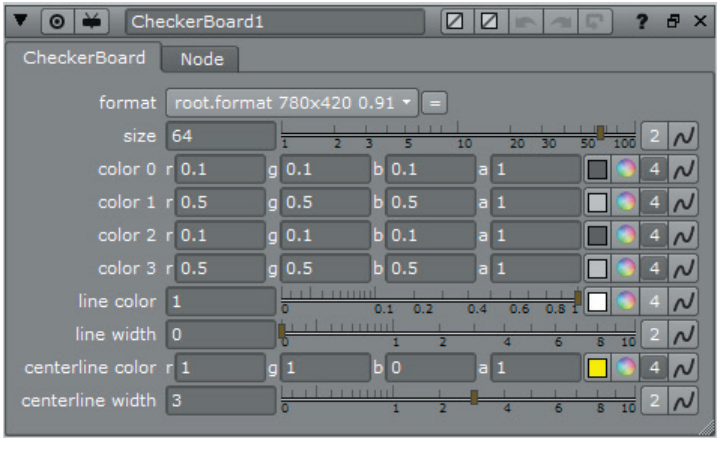


Figure 3-89 *The* ***CheckerBoard1*** *node properties panel*

2. In the **CheckerBoard** tab of the **CheckerBoard1** node properties panel, set the values of the parameters as given next:

size: **16** centerline width: **0**

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

3. Click on an empty area of the **Node Graph** panel and press M; the **Merge1** node is inserted in the **Node Graph** panel.

4. Press and hold the left mouse button on the **B** input of the **Merge1** node and then drag the cursor to the **CheckerBoard1** node; a connection between the **CheckerBoard1** and **Merge1** nodes is established.

5. Press and hold the left mouse button on the **A** input of the **Merge1** node and drag the cursor to the **Read1** node to establish a connection between the two nodes, refer to Figure 3-90. You will notice that an additional **A** input is created on the **Merge1** node.

   Next, you need to display the effect of the merge operation.

6. Select the **Merge1** node in the **Node Graph** panel and then press 1; the output is displayed in the **Viewer1** panel, as shown in Figure 3-91.

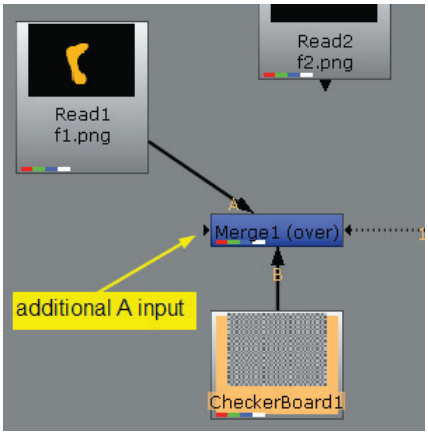


Figure 3-90 *The* ***Merge1*** *node connected to the* ***Read1*** *and* ***CheckerBoard1*** *nodes*

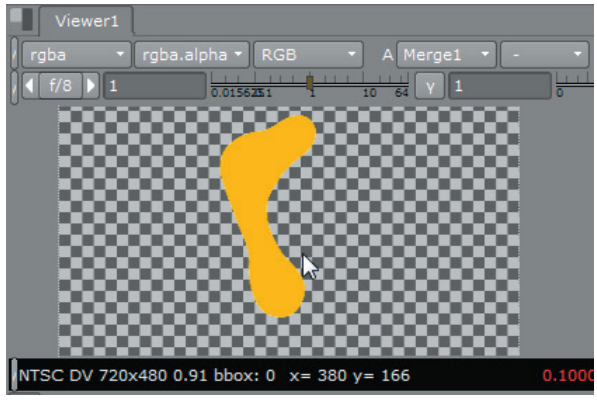


Figure 3-91 *The output of the* ***Merge1*** *node*

7. Similarly, connect all the remaining **Read#** nodes to the **Merge1** node, refer to Figure 3-92.

Rendering and Saving the Script

In this section, you will render the script using the **Write** node.

1. Make sure the **Merge1** node is selected in the **Node Graph** panel and then press W; the **Write1** node is connected to the **Merge1** node and its properties panel is displayed in the **Properties Bin**.

2. In the **Write** tab of the **Write1** node properties panel, click on the folder icon corresponding to the **file** parameter; the **Write1: Select file(s)** dialog box is displayed. In this dialog box, navigate to *\Documents\Nuke_Projects\c03_tut\Render* and then append **c03_tut1_rndr.jpeg** to the path displayed in the pathname field. Next, choose the **Save** button.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

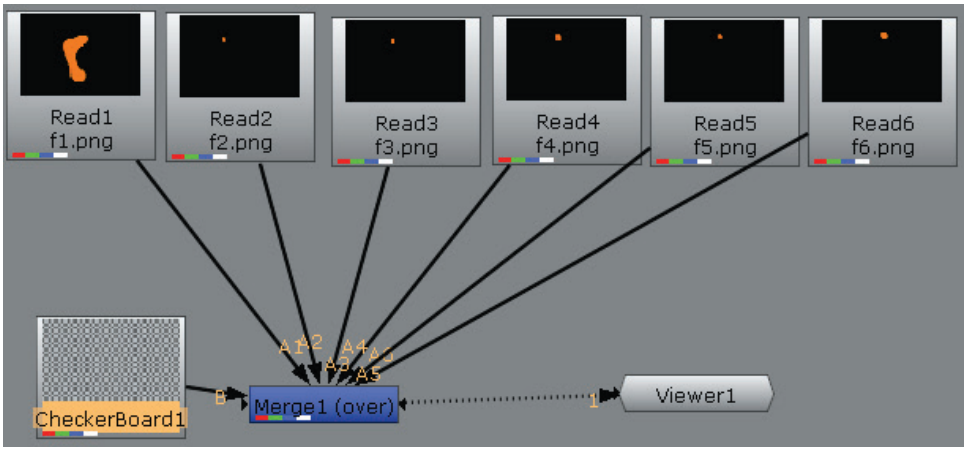


Figure 3-92 *All the* ***Read#*** *nodes connected to the* ***Merge1*** *node*

You will notice **jpeg** is automatically selected in the **file type** drop-down and some additional parameters are displayed in the **Write1** node properties panel. The additional file type specific parameters are displayed when you select different file types from the **file type** drop-down.

3. Enter **1** in the **quality** field. This will set the *jpeg* output quality to maximum.

4. Choose the **Render** button; the **Render** dialog box is displayed. In this dialog box, accept the default values and then choose the **OK** button; the **Progress** window is displayed. This window will be automatically closed on completion of the rendering process. Next, navigate to *\Documents\Nuke_Projects\c03_tut\Render* using Windows Explorer and then double-click on the *c03_tut1_rndr.jpeg* to view the rendered output.

5. Choose **File > Save** from the menu bar to save the script.

Tutorial 2

In this tutorial, you will apply various merge algorithms to *clouds.png* and *skyfog.jpeg*. Figures 3-93 and 3-94 show the *clouds.png* and *skyfog.jpeg*, respectively. Figures 3-95 and 3-96 show the result after using the **color-dodge** and **hard-light** algorithms, respectively.

(Expected time: 45 min)

The following steps are required to complete this tutorial:

a. Set the project.
b. Import images.
c. Merge images.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com



Figure 3-93 *The clouds.png image*



Figure 3-94 *The skyfog.jpeg image*

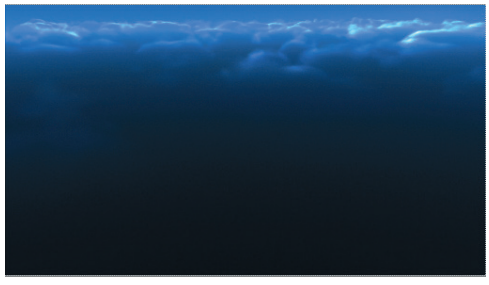


Figure 3-95 *The output of the* ***color-dodge*** *algorithm*



Figure 3-96 *The output of the* ***hard-light*** *algorithm*

Setting the Project

As the resolution of images used in this tutorial is PAL 768x576, you need to set the project accordingly. In this section, you will specify the project settings.

1. Choose **File > New** or press CTRL+N; a new NukeX screen is displayed.

2. Choose **File > Save** from the menu bar; the **Save script as** dialog box is displayed. In this dialog box, navigate to *\Documents\Nuke_Projects\c03_tut* and then append **c03_tut02** to the path displayed in the pathname field. Next, choose the **Save** button to save the script.

3. Hover the cursor over the **Node Graph** panel and press S; the **Project Settings** panel is displayed in the **Properties Bin**. Make sure the **Root** tab is chosen in it. Next, in the panel, select **new** from the **full size format** drop-down; the **New format** dialog box is displayed. In this dialog box, set the values of the fields as given below:

   name: **PAL768x576** file size w: **768** file size h: **576**
   pixel aspect: **1.333**

   Next, choose the **OK** button to close the dialog box. You will notice that new format created is displayed in the **full size format** drop-down as well as in the **Viewer1** panel.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Importing Images

In this section, you will import the *clouds.png* and *skyfog.jpeg* images in the script.

1. Choose the **Image** button from the **Nodes** toolbar; the **Image** menu is displayed. Next, choose **Read** from this menu; the **Read File(s)** dialog box is displayed. In this dialog box, choose **Home > Documents > Nuke_Projects > c03_tut > Media_Files > clouds.png** and then choose the **Open** button; the **Read1** node is inserted in the **Node Graph** panel.



Note

*In the **Read File(s)** dialog box, you can quickly jump to the Nuke_Projects folder by clicking on the Nuke_Projects bookmark created in Tutorial 1.*

2. Import *skyfog.jpeg* in the script as discussed above. Now, the **Read2** node is inserted in the **Node Graph** panel, as shown in Figure 3-97.

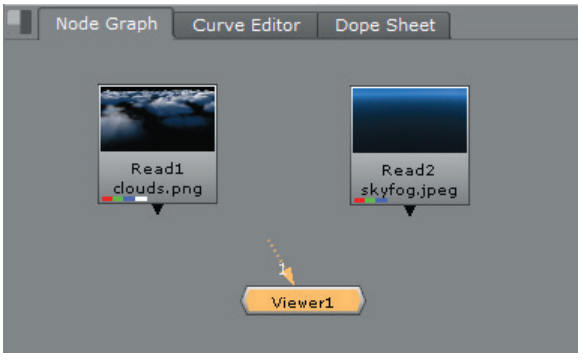


Figure 3-97 *The **Read#** nodes in the **Node Graph** panel*

Merging Images

In this section, you will create a composite using the imported images.

1. Select the **Read1** node in the **Node Graph** panel. Next, press and hold SHIFT and then select the **Read2** node and then press M; the **Merge1** node is inserted in the **Node Graph** panel and connections are established between the **Read** nodes and the **Merge1** node, refer to Figure 3-98.

   By default, the first node you selected in the **Node Graph** panel is connected to the **A** input of the **Merge** node whereas the second node is connected to the **B** input. You can swap the **A** and **B** inputs of a **Merge** node by first selecting it in the **Node Graph** panel and then pressing SHIFT+X.

2. Select the **Merge1** node in the **Node Graph** panel and press 1; the output of the merge operation is now displayed in the **Viewer1** panel.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

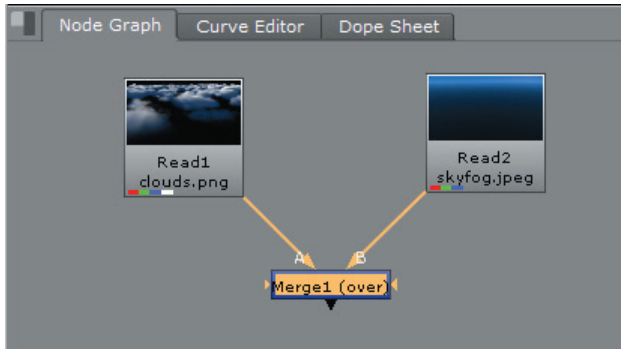


Figure 3-98 *The* ***Merge1*** *node in the* ***Node Graph*** *panel*

In the **Merge** tab of the **Merge1** node properties panel, **over** is selected by default in the **operation** drop-down, refer to Figure 3-99. It layers the **A** input over the **B** input according to the alpha of the **A** input. The formula used to combine pixels by the **over** algorithm is A+B(1-a). Figures 3-100 to 3-107 show the output of various merge algorithms.

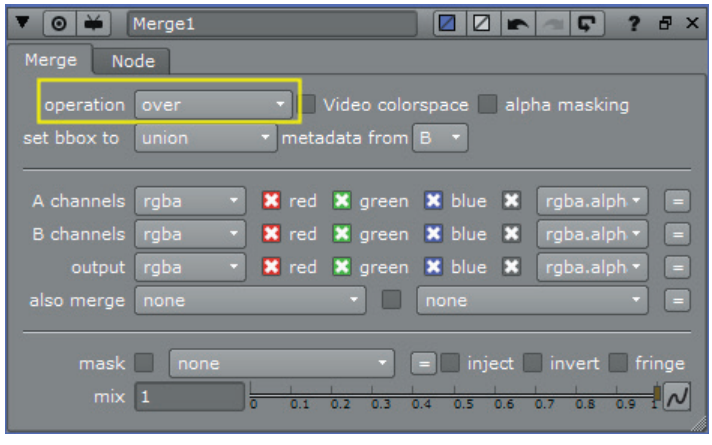


Figure 3-99 *The* ***Merge1*** *node properties panel*

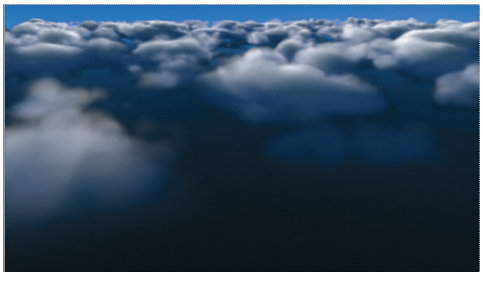


Figure 3-100 *The output of the* ***over*** *algorithm*

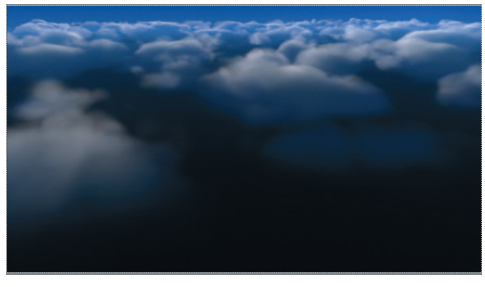


Figure 3-101 *The output of the* ***average*** *algorithm*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

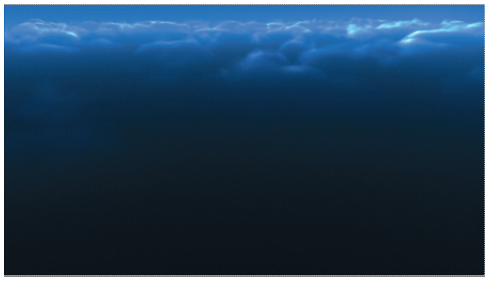


Figure 3-102 *The output of the* ***color-dodge*** *algorithm*

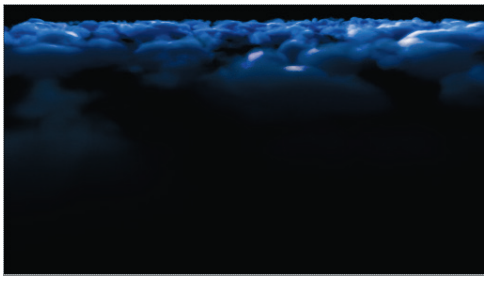


Figure 3-103 *The output of the* ***hard-light*** *algorithm*

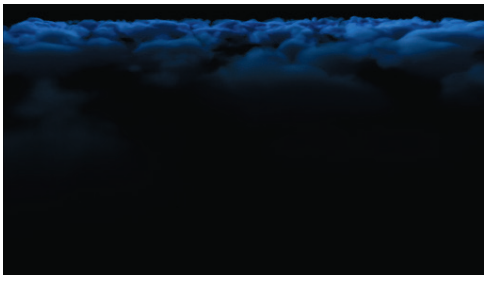


Figure 3-104 *The output of the* ***multiply*** *algorithm*



Figure 3-105 *The output of the* ***overlay*** *algorithm*

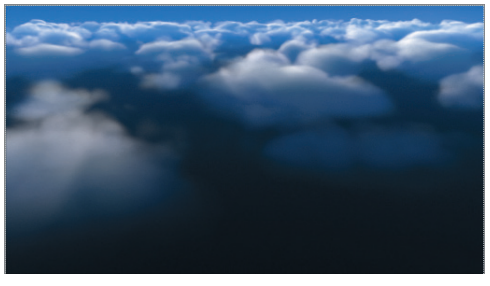


Figure 3-106 *The output of the* ***screen*** *algorithm*



Figure 3-107 *The output of the* ***soft-light*** *algorithm*

3. In the **Merge** tab of the **Merge1** node properties panel, select **soft-light** from the **operation** drop-down. Next, select the **Video colorspace** check box; colors are converted to default 8-bit colorspace before doing the composite and then the results are converted back to linear, refer to Figure 3-108.

4. Next, clear the **Video colorspace** check box.



Note

The alpha and other channels (except rgb) are merged without colorspace conversion. The default colorspace for monitor, 8-bit files, and 16-bit files is sRGB.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com



Figure 3-108 *The output of the* ***Merge1*** *node*

5. Select the **alpha masking** check box; the output of the merge operation is displayed according to the PDF/SVG spec, refer to Figures 3-109 and 3-110.



Figure 3-109 *The output of the merge operation before selecting the* ***alpha masking*** *check box*

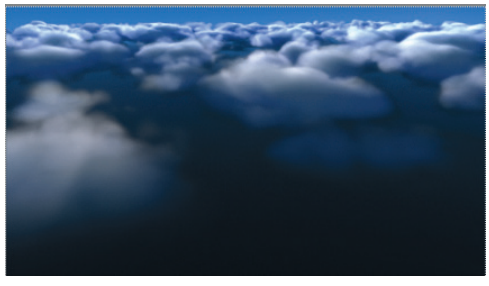


Figure 3-110 *The output of the merge operation after selecting the* ***alpha masking*** *check box*



Note

The ***alpha masking*** *check box is only enabled for those algorithms which make no difference to the output, for example, the* ***soft-light*** *algorithm.*

6. Enter **0.7** in the **mix** field.

   Next, you will modify the metadata.

7. Select the **Read2** node in the **Node Graph** panel and then press TAB; the TAB menu is displayed. Enter **crop** in the TAB menu field and then press ENTER; the **Crop1** node is inserted between the **Read2** and **Merge1** nodes, as shown in Figure 3-111.

8. In the **Crop** tab of the **Crop1** node properties panel, set the values, as shown in Figure 3-112.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

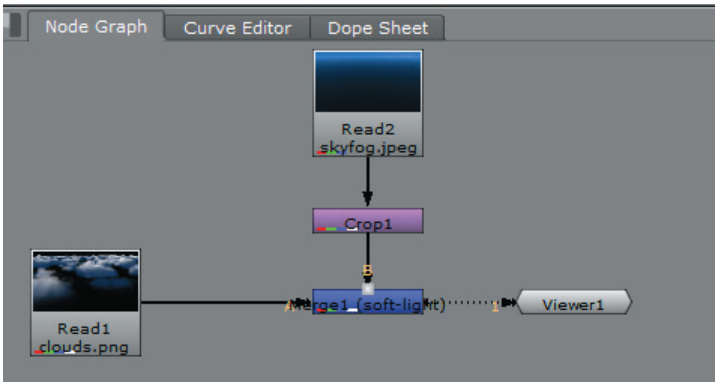


Figure 3-111 *The* ***Crop1*** *node inserted in the* ***Node Graph*** *panel*

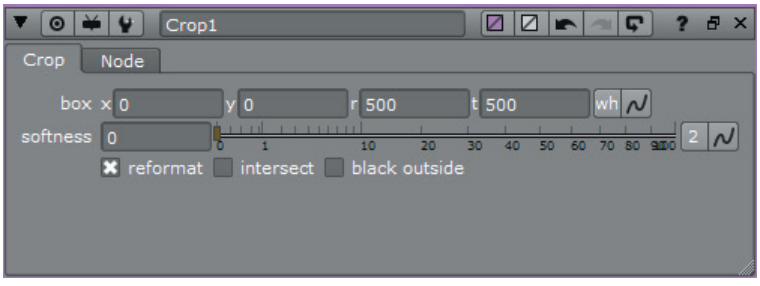


Figure 3-112 *The* ***Crop1*** *node properties panel*

9. In the **Merge** tab of the **Merge1** node properties panel, select **B** from the **set bbox to** and **metadata from** drop-downs. Figure 3-113 shows the new bounding box.

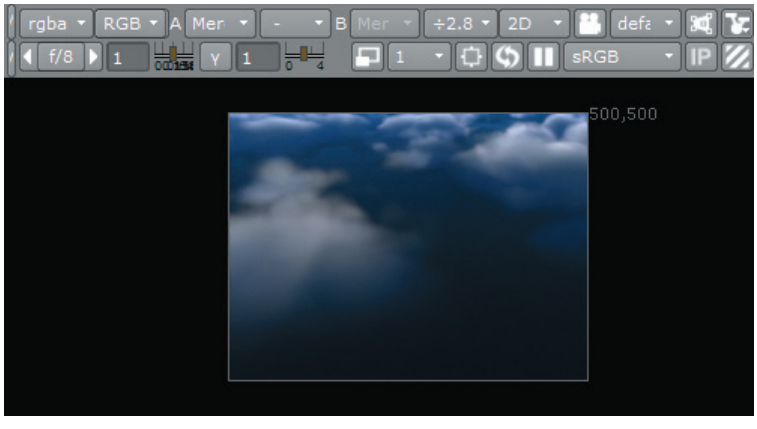


Figure 3-113 *The new bounding box displayed in the* ***Viewer1*** *panel*

Next, you will modify the metadata.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

10. Select the **Merge1** node in the **Node Graph** panel and then choose the **MetaData** button on the **Nodes** toolbar; the **MetaData** menu is displayed. Next, choose **ModifyMetaData** from the menu; the **ModifyMetaData1** node is inserted between the **Merge1** and **Viewer1** nodes, as shown in Figure 3-114.

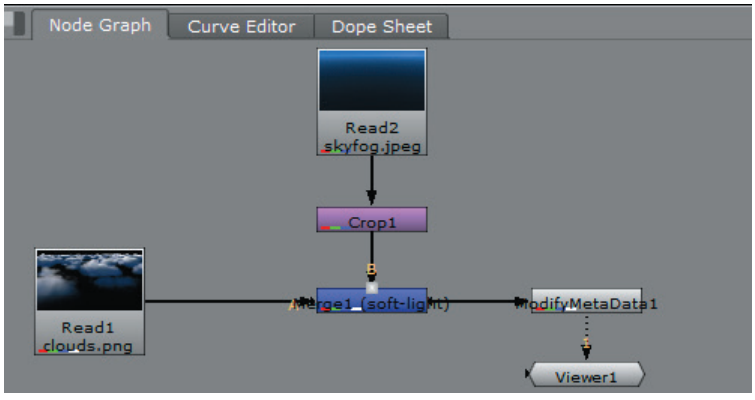


Figure 3-114 *The* ***ModifyMetaData1*** *node in the* ***Node Graph*** *panel*

11. In the **ModifyMetaData** tab of the **ModifyMetaData1** node properties panel, choose the **+** button; a placeholder is inserted in the **metadata** box.

12. Now, double-click on the first cell under the **key** column header; the **Pick metadata key** dialog box is displayed. In this dialog box, select **input/height** and then choose the **OK** button; the **input/height** key is displayed in the first cell under the **key** column header.

13. Double-click on the first cell under the **value** column header; the cell is converted into a field. Next, enter **500** in this field and then press ENTER, refer to Figure 3-115.

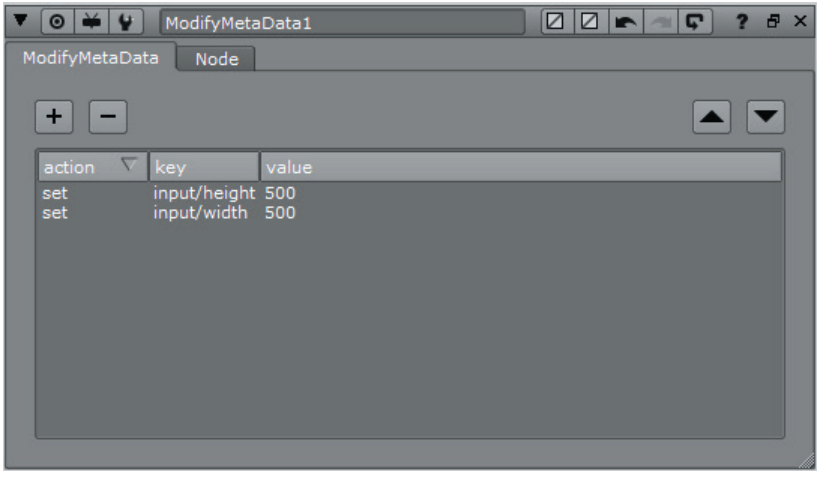


Figure 3-115 *Keys and values displayed in the* ***ModifyMetaData1*** *node properties panel*

14. Repeat the process described in steps 11 through 13 to add the **input/width** key in the metadata box and assign it a value of **500**, refer to Figure 3-115.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

15. Make sure the **ModifyMetaData1** node is selected in the **Node Graph** panel and then choose the **MetaData** button; the **MetaData** menu is displayed. Next, choose **ViewMetaData** from the menu; the **ViewMetaData1** node is inserted between the **ModifyMetaData1** and **Viewer1** nodes in the **Node Graph** panel.

    Note that the **ViewMetaData** tab of the **ViewMetaData1** node properties panel that the **input/height** and **input/width** keys display a value of 500, as shown in Figure 3-116.

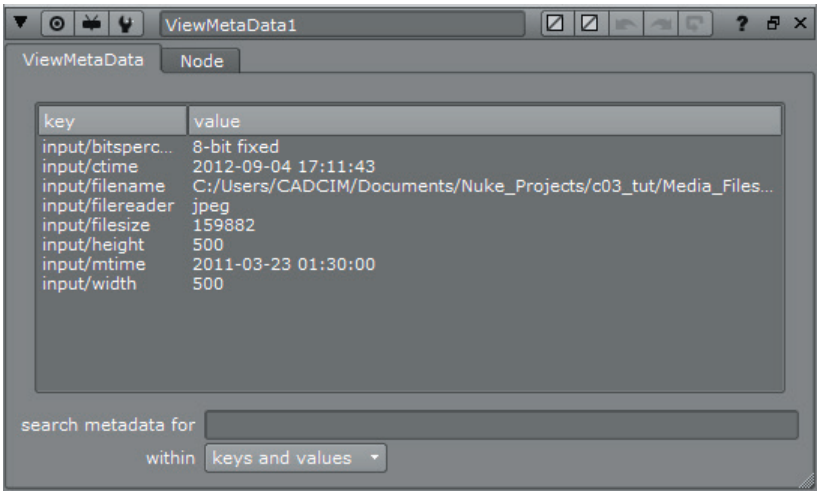


Figure 3-116 *The **ViewMetaData1** node properties panel*

16. Choose **File > Save** from the menu bar to save the script.

    Next, you need to render the script. For rendering, refer to Tutorial 1.

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. Which of the following shortcut keys is used to open the **Project Settings** panel when cursor is over the **Node Graph** panel?

    (a) M (b) K
    (c) S (d) R

2. Which of the following shortcut keys is used to add the **Grade** node to the **Node Graph** panel?

    (a) G (b) Y
    (c) T (d) N

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

3. To extract a node from a node network, choose _____ from the **Edit** menu.

4. To toggle between enable and disable modes of a node, press _____ .

5. The **Checkerboard** node is used as a _____ for a texture or a background.

6. The default colorspace for monitor, 8-bit files, and 16-bit files is _____ .

7. You can select all connected nodes in a node network by pressing _____.

8. You can rename a node in the **Node Graph** panel by pressing _____.

9. Press _____ to swap between the **A** and **B** inputs of a **Merge** node.

10. To select all nodes in a script, choose _____ from the menu bar.

11. To duplicate a selected node, press _____ .

12. Choose **Edit > Node > Disable/Enable** from the menu bar to enable or disable nodes. (T/F)

13. The options in the **metadata from** drop-down are used to specify which input's node data is passed down the node tree. (T/F)

14. If a node is selected in the **Node Graph** panel then on adding a new node, the new node will be automatically connected to the selected node. (T/F)

15. Wildcards cannot be used in a search string while searching a node by its name. (T/F)

16. You can select nodes based on their color, class, and label. (T/F)

Review Questions

Answer the following questions:

1. Which of the following shortcut keys is used to add the **ColorCorrect** node to the **Node Graph** panel?

(a) W (b) C
(c) E (d) V

2. Which of the following combinations of shortcut keys is used to remove the input connector of the selected node?

(a) CTRL+D (b) ALT+K
(c) ALT+M (d) CTRL+P

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

3. Which of the following combinations of shortcut keys is used to clone a node?

(a) ALT+D (b) ALT+K
(c) ALT+M (d) ALT +P

4. Which of the following combinations of shortcut keys is used to force clone a node?

(a) CTRL+ALT+D (b) CTRL+ALT+SHIFT+K
(c) ALT+SHIFT+M (d) SHIFT+K

5. Which of the following combinations of shortcut keys is used to group nodes?

(a) CTRL+G (b) CTRL+ALT+G
(c) ALT+G (d) SHIFT+G

6. Which of the following combinations of shortcut keys is used to expand a group?

(a) CTRL+G (b) CTRL+ALT+G
(c) ALT+SHIFT+G (d) SHIFT+G

7. To select multiple nodes, press and hold _____ while clicking on the nodes.

8. To select all upstream, nodes press and hold _____ while dragging on a node.

9. Most of the nodes have input, output, and mask connectors. (T/F)

10. You can enter HTML markup while creating a note using the **Sticky Note** node. (T/F)

11. Press and hold ALT and then click on the Close(x) button of any of the panels to close all the properties panels in the **Properties Bin**. (T/F)

12. The M key is used to add the **Merge** node in the **Node Graph** panel. (T/F)

13. The bounding box is used to speed up processing time. (T/F)

14. A node can be exported as a NukeX script. (T/F)

15. The **mix** field in the **Merge** node properties panel is used to mix the result of the **Merge** node and its input. (T/F)

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Answers to Self-Evaluation Test

1. c, **2.** a, **3. Extract**, **4.** D, **5.** placeholder, **6. sRGB**, **7.** CTRL+ALT+A, **8.** N, **9.** SHIFT+X, **10. Edit > Select All**, **11.** ALT+C, **12.** T, **13.** T, **14.** T, **15.** F, **16.** T