

Chapter 1

Exploring NukeX 7 Interface

Learning Objectives

After completing this chapter, you will be able to:

- *Start The Foundry NukeX*
- *Get familiar with the NukeX interface*
- *Understand the functions of nodes in NukeX*
- *Load, save, and close NukeX scripts*
- *Set projects for composition*
- *Load image sequences into NukeX*
- *Understand proxies*

INTRODUCTION

Welcome to the world of Nuke, an Academy Award® winning compositing software application. Nuke, a production-proven visual effects tool, is used to composite images, multiple stills, 3D renders, and image sequences into a artwork. Its state-of-the-art image processing technology delivers unparalleled speed on very complex compositing challenges. It has been used to create stunning images for feature films, such as Avatar, Alice in Wonderland, District 9, Transformers, Pirates of the Caribbean, Resident Evil, Quantum of Solace, and so on. Nuke comes in three versions: Nuke, NukeX, and Nuke PLE (Personal Learning Edition). NukeX has all features of Nuke with an integrated 3D camera tracker, depth map generator, projection solver, noise removal system, planar tracker, 3D particles system, automatic lens distortion correction tools, and FurnaceCore. The PLE version is free but the output on rendering and in the Viewer shows watermark placed on them. In this textbook, you will learn about the features and capability of NukeX in detail.

STARTING The Foundry NukeX 7

You can start NukeX from the taskbar or by using the shortcut icon placed on your desktop. To start NukeX from the taskbar, choose **Start > All Programs > The Foundry > Nuke 7.0v4 > NukeX 7.0v4**, as shown in Figure 1-1; two windows will be displayed on the computer screen, the **NukeX 7.0v4** command window and the **NukeX** interface window, as shown in Figures 1-2 and 1-3, respectively.

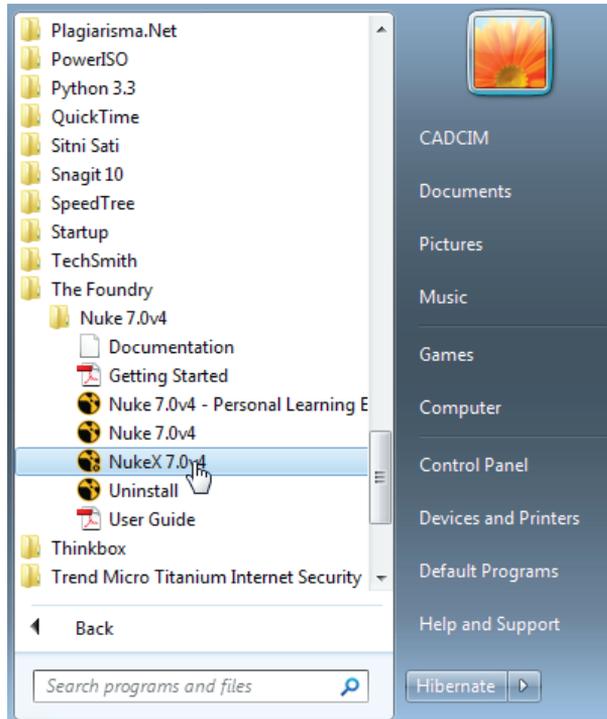


Figure 1-1 Starting NukeX 7 from the taskbar

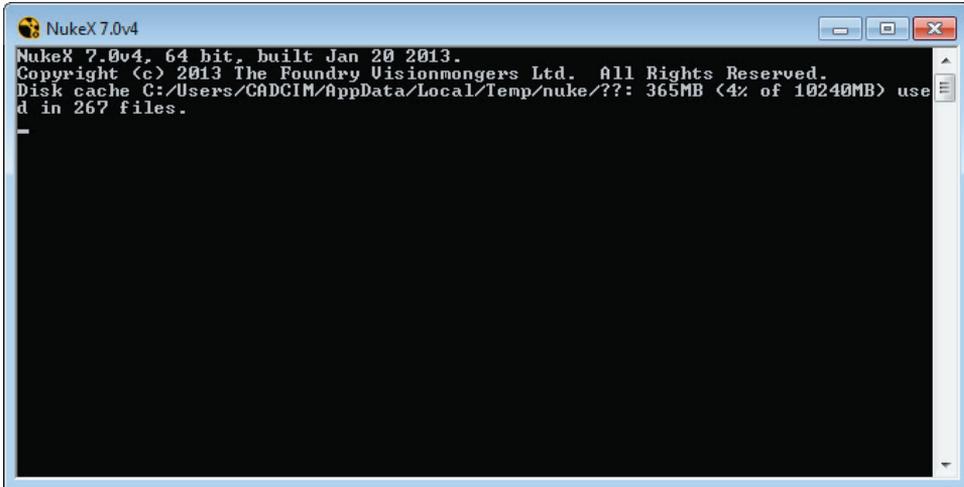


Figure 1-2 The NukeX 7.0v4 command window

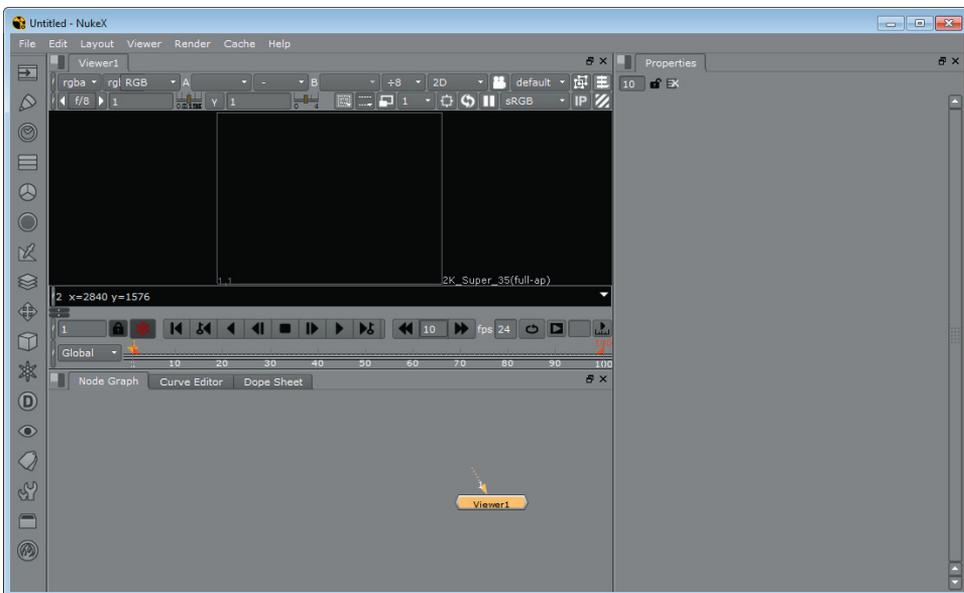


Figure 1-3 The default NukeX interface window

EXPLORING NukeX INTERFACE

The main interface window of NukeX consists of various components such as panes, panels, toolbar, and menu bar, refer to Figure 1-4. These components are discussed next.

Menu Bar

The menu bar is located at the top of the NukeX interface. The options in the **File** menu are used to perform disk operation commands and import scripts. The **Edit** menu contains editing commands, preferences, and project settings. The **Layout** menu contains options for saving

and accessing the layout schemes. The **Layout** menu also consists of options for showing the **Curve Editor** panel and the full screen toggle command. The options in the **Viewer** menu are used to create and connect viewers. The **Render** menu contains various render commands to render the output. The **Help** menu contains commands to access documentation, release notes, mailing lists, and tutorials. The options in the **Cache** menu are used to cache files as well as for clearing the disk cache and buffers.

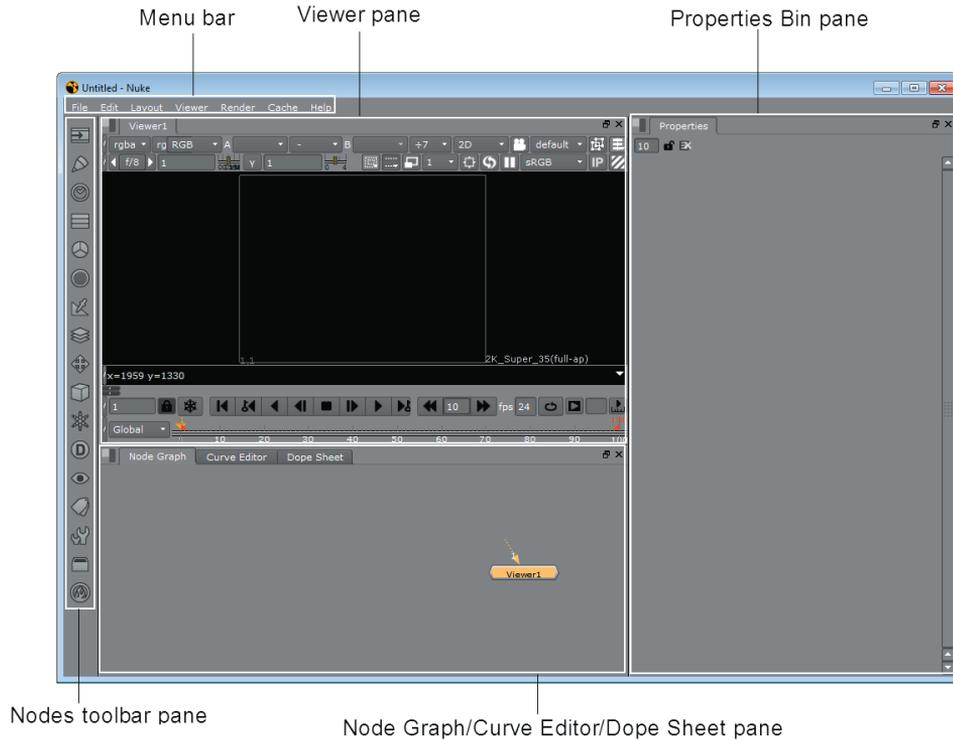


Figure 1-4 The various components of the NukeX interface

Panes

The default NukeX interface consists of **Nodes** toolbar pane, the **Viewer** pane, the **Node Graph/Curve Editor/Dope Sheet** pane and the **Properties Bin**. The tabs inside the panes are called panels. By default, the **Node Graph/Curve Editor/Dope Sheet** pane is displayed at the lower left corner of the NukeX interface, the **Viewer** pane is located at the upper left corner, and the **Properties Bin** pane at the right. The **Nodes** toolbar pane is located on the left of the NukeX interface and the menu bar is located at its top. The options in the **Nodes** toolbar are used to add nodes to the **Node Graph** panel. The **Node Graph** panel is used to build a network of nodes to create a composite. The **Node Graph** is also known as DAG (Directed Acyclic Graph). The **Curve Editor** and **Dope Sheet** panels are used to refine the animation curves and keyframes. The **Properties Bin** pane is used to adjust the controls of the nodes. The **Viewer** pane is used to preview the output of the nodes. You can also add the **Script Editor** and **Progress Bar** panels to the panes. The **Script Editor** panel is used to execute the Python commands, whereas the **Progress Bar** panel is used to display the progress bars. The process of adding a panel to a pane will be discussed later in this chapter.

You can modify the position and size of the panes. You can also add or remove panels from the panes. Various methods to modify the arrangement of the panes and panels are discussed next.

Changing the Size of a Pane

To change the size of a pane, place the cursor on the divider line between the panes; the shape of the cursor will change, as shown in Figure 1-5. Next, drag the cursor to resize the pane as required and then release the mouse button.



Figure 1-5 The changed shape of the cursor

Splitting a Pane

To split a pane, choose the Checkerboard button located at the upper left corner of the pane; the Content menu will be displayed, as shown in Figure 1-6. Next, choose **Split Vertical** or **Split Horizontal** from the menu to split the pane vertically or horizontally. Figure 1-7 shows the horizontally split **Node Graph/Graph Editor/Dope Sheet** pane. To restore the default layout, press SHIFT+F1.

Adding Panel to a Pane

To add a new panel to a pane, choose the Checkerboard button and then choose the desired option such as **Script Editor** or **New Viewer** from the Content menu displayed.



Tip: To cycle through the panels in a pane, use the CTRL+T keys.

Closing a Panel

To close a panel, click on the Close (x) button located on the top right corner of the panel or choose the **Close Tab** option from the Content menu.

Floating a Pane or Panel

To float a pane or a panel, choose the **Float Pane** or **Float Tab** option from the Content menu.



Tip: You can dock or undock the panels from the NukeX interface. To undock a panel, press CTRL and then click on the label of the panel name; a floating window will be displayed. Alternatively, you can drag the panel outside the borders of the main NukeX window. To dock the floating panel back to the pane, drag the panel name label over the pane; an orange line will appear at the top of the pane and then release the mouse button to dock the panel back to the pane.

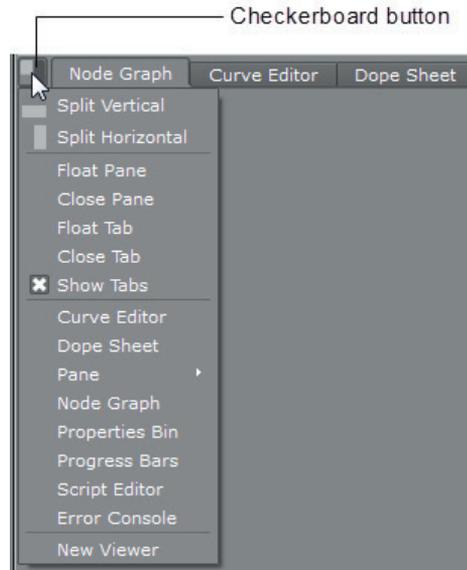


Figure 1-6 The Content menu displayed on choosing the Checkerboard button

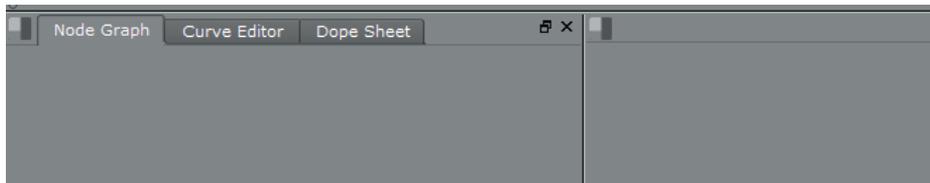


Figure 1-7 The Node Graph/Curve Editor/Dope Sheet pane split horizontally



Note

In NukeX, you can save and restore upto 6 layouts. To save a layout, choose **Layout > Save Layout #** from the menu bar. To restore a layout, choose **Layout > Restore Layout #** from the menu bar. Here, the # symbol represents the layout number.

Nodes Toolbar

The **Nodes** toolbar is located on the left of the NukeX interface. All nodes can be accessed from it. By default, this toolbar contains 17 buttons. When you choose a button from this toolbar, a menu containing a list of nodes associated with that button will be displayed. The functions of these buttons and their corresponding nodes are discussed next.

Image



This button is used to access the **Read** and **Write** nodes, miscellaneous NukeX elements and **Viewer** nodes. Brief description of these nodes is given next.

Read: This node is used to load an image from the disk.

Write: This node is used to write the output of all upstream nodes connected to it.

UDIM Import: This node is used to import a set of texture patches that follow the UDIM numbering scheme.

Constant: This node is used to produce a constant color image.

CheckerBoard: This node is used to generate a checkerboard image which can be used as a placeholder for a texture or background.

ColorBars: This node is used to generate the SMPTE (Society of Motion Picture and Television Engineers) color bar test pattern.

ColorWheel: This node is used to generate a color wheel image.

CurveTool: This node is used to create an animation curve. This curve can be used to control the size and position of the black areas, exposure changes, and luminance in a sequence.

Viewer: This node is used to display the render output of the process node in the Viewer panel.



Draw

This button is used to access the rotoscopy, grain, lens flare, and vector based nodes. The nodes under this button are discussed next.

Roto: This node is specifically used for rotoscopy. It is used to create Bezier and B-Spline roto shapes.

RotoPaint: This node is similar to the **Roto** node with the only difference that by using the **RotoPaint** node, you can perform tasks such as rig removal, dustbusting, and garbage removal.

Dither: This node is used to add random noise to an image to hide the quantization effects.

DustBust: This node is used to remove dust or scratches from the images. It clones multiple areas from a source to a destination.

Grain: This node is used for adding synthetic grain to a footage.

ScannedGrain: This node is used to apply the actual scans of the film grain to the input image to replicate the grain of the scanned film stock.

Glint: This node is used to create star-shaped points around the brighter points in an image.

Grid: This node is used to draw a grid of horizontal and vertical lines.

Flare: This node is used to simulate the lens flare effect.

LightWrap: This node is used to wrap the light from the background onto the foreground object.

MarkerRemoval: This node is used to remove tracking markers from a footage by using the roto shapes.

Noise: This node is used to apply various types of seamless noise to the image, based on the perlin noise algorithm.

Radial: This node is used to generate a radial gradation ramp.

Ramp: This node is used to generate a gradient ramp. However, its usage is obsolete now.

Rectangle: This node is used to generate a solid color rectangle shape.

Sparkles: This node is used to generate sparkles or rays.

Text: This node is used to add text overlays on the images.

Time



This button is used to access time management nodes. The nodes under this button are discussed next.

Add 3:2 pulldown: This node is used to convert a 24fps film to 29.97fps interlaced video.

Remove 3:2 pulldown: This node is used to remove a 3:2 pull-down from the input image.

AppendClip: This node is used to append one clip to another. It can be used to join the clips, add transitions between clips, and slip edit the clips.

FrameBlend: This node is used to interpolate frames by blending the frame that precede and follow it.

FrameHold: This node is used to return a constant frame from the input. If the frame increment is set to a non zero values, this node will return every nth frame.

FrameRange: This node is used to set a range of frames.

Kronos: This node is a retime node and used to slow down or speed up the image sequence.

OFlow: This node is used to slow down or speedup the input footage. Additionally, you can add the motion blur to an image or enhance the motion blur already applied to an image.

Retime: This node is used to change the timing of the input clip. You can use this node to slow down, speed up, and reverse frames in an input image.

TemporalMedian: This node is used to remove grain from the footage by calculating the median of the current frame, the frame before, and the frame after the current frame.

TimeBlur: This node is used to generate temporal motion blur.

NoTimeBlur: This node is used to avoid unwanted computations in blur operations.

TimeEcho: This node is used to generate echo like effects by merging a range of frames into a single frame.

TimeOffset: This node is used to move the input clip forward or backward in time.

TimeWarp: This node is used to change the timing of the input clip.

TimeClip: This node is used to move the input image forward or backward in time.

VectorGenerator: This node is used to produce images containing motion vector field.

Channel



This button is used to access channel management nodes. The nodes under this button are discussed next.

Shuffle: This node is used to rearrange up to eight channels in a data stream.

ShuffleCopy: This node is used to reorder up to eight channels from two inputs. Also, it is used to copy channels between data streams.

Copy: This node is used to replace channels in the **B** input with channels from the **A** input.

ChannelMerge: This node is used to merge two channels. The result of the merge operation is saved in the selected output channel.

Add: This node is used to add a channel to an image.

Remove: This node is used to remove channels from an image.

Color



This button is used to access color correction nodes. The nodes under this button are discussed next.

Math (Add): This node is used to offset values of a channel.

Math (Multiply): This node is used to multiply values of a channel by a factor.

Math (Gamma): This node is used to apply a constant gamma value to a set of channels.

Math (ClipTest): This node is used to create zebra stripes for the colors that are outside the specified range.

Math (ColorMatrix): This node is used to transform a color channel. It multiplies the RGB colors by an arbitrary 3x3 matrix.

Math (Expression): This node is used to reference pixels in other channels by using C like syntax expressions.

OCIO (OCIO CDLTransform): This node is used to apply an ASC CDL (American Society of Cinematographers Colors Decesion List) grade based on the OpenColorIO library.

OCIO (OCIO ColorSpace): This node is similar to the regular **ColorSpace** node but it is based on the OpenColorIO library.

OCIO (OCIO Display): This node is used to apply a colorspace conversion to an image sequence.

OCIO (OCIO FileTransform): This node is used to load a colorspace conversion from a file and then apply it using the OpenColorIO library.

OCIO (OCIO LogConvert): This node is used to convert compositing log data to scene linear data.

3D LUT (CMSTestPattern): This node is used to generate calibration patterns for color management.

3D LUT (GenerateLUT): This node is used to generate color management lookup tables.

3D LUT (Vectorfield - Apply 3D LUT): This node is used to transform the input colors based on the 3D vectors.

Clamp: This node is used to clamp all channels to the 0-1 range.

ColorLookup: This node is used to make gamma, gain, and contrast adjustments to an input image using the lookup tables.

Colorspace: This node is used to convert images from one color space into another.

ColorTransfer: This node is used to change the color of an image to match the color of another image.

ColorCorrect: This node is used to adjust the ranges of shadows, midtones, and highlights of an input image.

Crosstalk: This node is used to add or remove crosstalk or bleeding between color channels.

Exposure: This node is used to adjust the exposure of the input footage.

Grade: This node is used to define the black and white points by sampling pixels from the Viewer.

Histogram: This node is used to plot the incoming image as a histogram. It also allows tonal range corrections.

HistEQ: This node is used to equalize the histogram of an image.

HueCorrect: This node is used to make precision adjustments to the levels of saturation in the range of hues.

HueShift: This node is used to transform the colorspace of an image by using the CIE XYZ colorspace.

HSVTool: This node is used to adjust the HSV components of an input image.

Invert: This node is used to invert values of a channels.

Log2Lin: This node is used to perform the log2lin or lin2log conversion used by Cineon files.

PLogLin: This node is used to convert the logarithmic color space to linear color space based on a single gray point.

MinColor: This node is used to calculate the difference between the darkest value in the input image and the target value, thus, the darkest pixel becomes the value.

Posterize: This node is used to reduce the color resolution of an image.

RollOffContrast: This node is used to implement a contrast curve with a smooth falloff equally across all channels.

Saturation: This node is used to change the saturation of the incoming image data.

Sampler: This node is used to convert the scanline of pixel data into a lookup curve.

SoftClip: This node is used to compress the high dynamic range imagery into the 0-1 range.

Toe: This node is used to lift the black levels without much affecting the whites in the input image.

TrueLight: This node is used to preview the final look of the film on the monitor display. This technology was developed by Filmlight.

Filter



This button is used to access the blur and sharpen nodes. The nodes under this button are discussed next.

Blur: This node is used to apply blur to an image using various filter algorithms.

Bilateral: This node is used to smoothen the images while preserving their edges.

BumpBoss: This node is used to produce an emboss effect by overlapping the **A** input with the **B** input.

Convolve: This node is used to create custom filter effects using the user specified filter images.

Defocus: This node is used to apply the defocus effect on an image using the disc filter.

DegrainBlue: This node is used to reduce noise in the blue channel.

DegrainSimple: This node is used to reduce noise in the red, green, and blue channels.

DirBlur: This node is used to apply various types of directional blur effects.

Denoise: This node is used to remove noise from the input image without losing image quality.

EdgeBlur: This node is used to apply blur to the edges within the matte.

EdgeDetect: This node is used to detect the edges of an image and blur them.

Emboss: This node is used to produce the emboss effect by offsetting the original input.

Erode (fast): This node is used to erode or dilate RGBA channels. It applies the Box Morphological filter.

Erode (filter): This node is similar to the Erode(fast) but it is computational intensive. You can improve the erode quality using the options available in the **filter** drop-down.

Erode (blur): This node is used to erode or dilate RGBA channels by using blur.

Glow: This node is used to generate the glow effect.

GodRays: This node is used to produce godrays, lighting, and blur effects.

Laplacian: This node is used to blur the image and then subtract the original image from the blurred image.

LevelSet: This node is used to create high quality dilate or unpremultiply operations.

Matrix: This node is used to apply a custom convolution matrix onto an image.

Median: This node is used to remove the single pixel noise without compromising the quality of the input image.

MotionBlur: This node is used to add blur based on the movement of the object in the scene.

MotionBlur2D: This node is used to output UV vectors for the **VectorBlur** node.

MotionBlur3D: This node is used to output the UV vectors that can be used to run the vector blur operator to produce multi-blur. It is specifically used for camera moves.

Sharpen: This node is used to sharpen the specific channels using the laplacian.

Soften: This node is used to produce the same effect as that of the **Sharpen** node, with the only difference being that the Laplacian effect is added to the original.

VectorBlur: This node is used to generate the motion blur by blurring each pixel into a straight line.

VolumeRays: This node is used to create ray lighting effects.

ZSlice: This node is used to select a slice from the input image by using the Z-Depth channel.

ZDefocus: This node is used to blur the image using the depth map channel.

Keyer



This button is used to access the matte extracting nodes. The nodes under this button are discussed next.

Difference: This node is used to produce the difference between two images as a matte.

HueKeyer: This node is used to calculate the hue of the incoming image and then uses the amount lookup curve to define the value to output in the alpha channel.

IBKGizmo: This node is used to generate matte by using the processed screen image (clean plate).

IBKColour: This node is used to process the screen image to be used with the **IBKGizmo** node.

Keyer: This node is used to pull the key from an image. It is a simple keyer.

Primatte: This node is used to pull the key from an image using the 3D RGB color space.

Keylight: This keyer is an advanced blue and green screen keyer.

Ultimate: This node is used to pull the key from an image. It is an advanced keyer developed by Ultimate Corporation. It preserves fine details such as hair and smoke.

Merge



This button is used to access various merge nodes. The nodes under this button are discussed next.

AddMix: This node uses the alpha of the **A** input to index two color correction lookup curves. The first input is used to multiply the **A** input, the second input is used to multiply the **B** input, and the results are added together.

KeyMix: This node is used to layer two images using a roto shape or an image as mask.

ContactSheet: This node is used to generate a contact sheet for all its inputs.

CopyBBox: This node is used to copy the bounding box from the **A** input onto the **B** stream.

CopyRectangle: This node copies a rectangle from the **A** input. It can be used to limit an effect to a specific area of the image.

Dissolve: This node is used to cross dissolve images with the help of weighted average of the two inputs.

LayerContactSheet: This node is used to generate a contact sheet from all channel sets.

Merge: This node is used for a pixel-by-pixel merge operation between input images.

Merges (Plus): This node is similar to the **Merge** node. In this node, the **operation** option is set to **plus** by default. It merges images together using the **A+B** compositing algorithm.

Merges (Matte): This node is similar to the **Merge** node. In this node, the **operation** option is set to **matte** by default.

Merges (Multiply): This node is similar to the **Merge** node. In this node, the **operation** option is set to **multiply** by default. It merges images together using the **AB, A if A < 0 and B < 0** compositing algorithm.

Merges (In): This node is similar to the **Merge** node. In this node, the **operation** option is set to **in** by default.

Merges (Out): This node is similar to the **Merge** node. In this node, the **operation** option is set to **out** by default. It merges images together using the **A(1-b)** compositing algorithm where **b** is the alpha value from the **B** input.

Merges (Screen): This node is similar to the **Merge** node. In this node, the **operation** option is set to **screen** by default. It merges images together using the **A+B-AB if A or B ≤ 1, otherwise max(A,B)** compositing algorithm.

Merges (Max): This node is similar to the **Merge** node. In this node, the **operation** option is set to **max** by default.

Merges (Min): This node is similar to the **Merge** node. In this node, the **operation** option is set to **min** by default. It merges images together using the **min(A,B)** compositing algorithm.

Merges (Absminus): This node is similar to the **Merge** node. In this node, the **operation** option is set to **difference** by default. It merges images together using the **abs(A-B)** compositing algorithm.

Merge Expression: This node is used to perform mathematical operations on channels using TCL expressions.

Switch: This node is used to switch between any numbers of inputs.

TimeDissolve: This node is used to dissolve two input data streams, starting the dissolve at the in frame and ending at the out frame.

Premult: This node is used to multiply all channels by the alpha channel.

UnPremult: This node is used to divide all channels by the alpha channel.

Blend: This node is used to blend images by creating weighted average of all inputs.

ZMerge: This node is used to merge images based on the Z channel.

Transform



This button is used to access transformation nodes. The nodes under this button are discussed next.

Transform: This node is used to reposition, scale, and move an image in the 2D space.

TransformMasked: This node is similar to the **Transform** node. However, it also provides controls for assigning a mask to protect certain areas of the frame from being transformed.

Card3D: This node is used to transform an image as it was printed on a flat card and placed in front of the camera. It gives you an additional z-axis for transformations that the **Transform** node does not provide.

AdjustBBox: This node is used to expand or crop the edges of the bounding box.

BlackOutside: This node is used to add black pixels outside an edge to the bounding box.

CameraShake: This node is used to add random camera shake including the motion blur.

Crop: This node is used to crop an image.

CornerPin: This node is used to map the four corners of an image sequence using the information taken from the tracking data.

SphericalTransform: This node is used to convert an environment map into another type of environment map from the HDR images.

IDistort: This node is used to distort an image based on UV channels.

LensDistortion: This node is used to apply or remove simple lens distortion. This node is only available in NukeX.

Mirror: This node is used to flip the image around the center of the format image area.

Position: This node is used to move the input by an integer number of pixels.

Reformat: This node is used to convert one image format (width/height) into another image format.

Reconcile3D: This node is used to project a user-specified 3D point through a connected camera onto the screen space.

Points3D: This node is used to extract 3D coordinates of 2D points.

PlanarTracker: This node is used to track surfaces that lie in a plane on the input image. It is only available in NukeX.

Tracker: This node is used to track a 2D motion with the in-built match-move functions.

TVIScale: This node is used to scale an image by a factor of two.

GridWarp: This node is used to transfer image information from a bezier grid to another bezier grid.

SplineWarp: This node is used to transform a source image from its original shape into a deformed image using multiple Bezier or B-spline curves.

Stabilize: This node is used to remove the unwanted camera movement in the image sequence.

STMap: This node is used to move pixels in an image.

Tile: This node is used to create the titled copies of the input image.

3D



This button is used to access 3D compositing nodes. The nodes under this button are discussed next.

Axis: This node is used to define a 3D transformation axis to the scene to which you can parent other objects.

Geometry (Card): This node is used to create a plane.

Geometry (Cube): This node is used to generate a 3D cube.

Geometry (Cylinder): This node is used to generate a 3D cylinder.

Geometry (DepthToPoints): This node is a gizmo that contains the **DepthToPosition** and **PositionToPoints** nodes. It is used to generate a 3D point cloud using a depth pass and 3D camera.

Geometry (ModelBuilder): This node is used to create 3D models from 2D images.

Geometry (PointCloudGenerator): This node is used to analyze an input image sequence and generate a point cloud using the **CameraTracker** node. This node is only available in NukeX.

Geometry (PositionToPoints): This node is used to generate a 3D point cloud from an input image, such as a position pass.

Geometry (PoissonMesh): This node is used to generate a mesh using a dense point cloud. This node is only available in NukeX.

Geometry (Sphere): This node is used to generate a 3D sphere.

Geometry (ReadGeo): This node is used to import a geometry.

Geometry (WriteGeo): This node is used to write the geometry to an external file.

Lights (Light): This node is used to create light and it has three light types: point, directional, and spot.

Lights (Point): This node is used to create a point light that emits light in all directions.

Lights (Direct): This node is used to emit light in one direction.

Lights (Spot): This node is a point in 3D space that emits a cone-shaped light in a given direction.

Lights (Environment): This node is used to produce light from the HDR image.

Lights (Relight): This node is used to relight an image using the 3D lights if the image contains the normal and point position passes.

Modify (TransformGeo): This node is used to transform the input image in 3D space.

Modify (MergeGeo): This node is used to merge all input geometries into one geometry.

Modify (CrosstalkGeo): This node is used to offer direct global control over each of the vertices on geometry using the crosstalk curves.

Modify (DisplaceGeo): This node is used to modify the shape of the geometry based on an image.

Modify (GeoSelect): This node is used to select individual vertices on a 3D object.

Modify (LookupGeo): This node is used to modify the shape of the 3D geometry using the lookup curves.

Modify (LogGeo): This node moves the XYZ position of the points by raising a value to a power.

Modify (Normals): This node is used to adjust the normals of the object.

Modify (ProceduralNoise): This node is used to modify objects using Perlin noise.

Modify (RadialDistort): This node is used to warp the points to create a pin-cushion like distortion.

Modify (Trilinear): This node warps points by using the trilinear interpolation lattice.

Modify (UVProject): This node is used to specify the UV coordinates for an object.

Modify (RenderMan-ModifyRIB): This node is used to insert RIB statements into the script.

Shader (ApplyMaterial): This node is used to apply a material/shader to a geometry by using the **mat** input.

Shader (BasicMaterial): This node is used to apply the basic emissive, diffuse, and specular materials to the geometry.

Shader (Displacement): This node is used to apply displacement mapping. It adds geometrical details to the surface of an object.

Shader (FillMat): This node is used to add a constant color into the selected material channels.

Shader (MergeMat): This node is used to combine the multiple project 3D shaders.

Shader (BlendMat): This node is used to blend the pixels colored by the material with the background pixels.

Shader (Project3D): This node is used to project an input image through camera onto the 3D object.

Shader (Diffuse): This node is used to shade an image using the diffuse lambertian lighting calculations.

Shader (Emission): This node is used to simulate emissive lighting such as light emitting from a lamp.

Shader (Phong): This node is used to create accurate shading and highlights using the phong shading algorithm.

Shader (Specular): This node is used to apply specular lighting to a selected color layer.

Shader (Displacement): This node is similar to the **DisplaceGeo** node but it only displaces those parts of the geometry which are visible at any given moment, thus saving the render time.

Shader (UVTile): This node is used to modify the coordinates of a UV patch while projecting textures onto an object.

Shader (RenderMan - Reflection): This node is used to apply reflection to a surface.

Shader (RenderMan - Refraction): This node is used to apply refraction to a surface.

Camera: This node is used to define a projection to be used by a 3D renderer. It adds a camera to the scene.

CameraTracker: This node is used to create a virtual camera whose movement is same as that of the original camera.

DepthGenerator: This node is used to generate a depth map from the input image.

DepthToPosition: This node is used to create a position pass from a depth map and 3D camera.

ProjectionSolver: This node is used to solve a camera for a set of known 3D points. This node is only available in NukeX.

Scene: This node is used to group more than one piece of geometry, lights, and camera.

ScanlineRender: This node is used to render a scene.

RenderMan (PrmanRender): This node works with Pixar's RenderMan to generate quality output. This node is only available in NukeX.

Particles



This button is used to access the particles nodes. The nodes under this button are discussed next.

ParticleEmitter: This node is used to emit particles.

ParticleBounce: This node is used to make 3D particles bounce off a 3D shape.

ParticleCurve: This node is used to apply a curve to particle controls such as size or mass.

ParticleDirectionalForce: This node is used to apply a directional force to the particles.

ParticleDrag: This node is used to apply drag to the particles to change their speed gradually.

ParticleExpression: This node is used to adjust the properties of the particles using expressions.

ParticleMerge: This node is used to merge multiple particle sets into one stream.

ParticleMotionAlign: This node is used to align a particle stream to the direction of their motion.

ParticleGravity: This node is used to apply gravity to the particles.

ParticleLookAt: This node is used to create a 3D point on which all particles look at the specified range.

ParticlePointForce: This node is used to attract or repel the particles to form a point in 3D space.

ParticleSpeedLimit: This node is used to restrict particles to a specified minimum and maximum speed.

ParticleSpawn: This node is used to emit particles from other particles.

ParticleTurbulence: This node is used to apply noise to the particle movement, thus dispersing the particles in 3D space.

ParticleVortex: This node is used to apply a circular force to the particles to attract them to an imaginary line.

ParticleWind: This node is used to apply a wind force to the particles.

ParticleSettings: This node is used to control the number of calculation steps taking place per animation frame.

ParticleToGeo: This node is used to create a 3D geometry for particles in the specified channels.

Deep



This button is used to access the deep pixels nodes. The nodes under this button are discussed next.

DeepColorCorrect: This node is used to color correct the deep images.

DeepCrop: This node is used as a regular **Crop** node to crop the deep images inside or outside the crop box. Also, it can be used to crop the deep data in front of or behind certain planes in depth.

DeepExpression: This node is used to apply TCL expressions to the deep data.

DeepFromFrames: This node is used to copy multiple frames from the input 2D image to sample a single deep frame.

DeepFromImage: This node is used to convert a standard image to a deep image.

DeepHoldout: This node is used to remove or fade out samples in the main input that are occluded by samples in the holdout input.

DeepMerge: This node is used to merge samples from multiple deep images.

DeepRead: This node is used to load deep images from the disk.

DeepRecolor: This node is used to merge the **color** input and the **depth** input.

DeepReformat: This node is similar to the **Reformat** node for deep images.

DeepSample: This node is used to pick samples from the deep image.

DeepToImage: This node is used to flatten a deep image.

DeepToPoints: This node is used to transform the deep pixel samples into points in 3D space, so that a point cloud is formed.

DeepTransform: This node is used to reposition the deep data.

DeepWrite: This node is used to write deep data to files.

Views



This button is used to access the views and stereoscopic view nodes. The nodes under this button are discussed next.

Stereo (Anaglyph): This node is used to make an anaglyph image out of inputs.

Stereo (MixViews): This node is used to mix two views together.

Stereo (SideBySide): This node is used to put the left and right views of the input next to each other.

Stereo (ReConverge): This node is used to shift convergence so that any selected point appears at screen depth when viewed with 3D glasses.

JoinViews: This node is used to join the left and right views to make stereo output.

OneView: This node is used to take one view from input for processing.

ShuffleViews: This node is used to swap the left and right views in the script.

Split and Join: This node can be used to split the view to apply processing to one view only. You can join them together later on.

MetaData



This button is used to access the metadata management nodes. The nodes under this button are discussed next.

ViewMetaData: This node is used to view the metadata of input.

CompareMetaData: This node is used to compare the metadata of two inputs.

ModifyMetaData: This node is used to alter the metadata being passed through the tree.

CopyMetaData: This node is used to copy metadata from one image to another.

AddTimeCode: This node is used to add a timecode to metadata.

ToolSets



This button is used to access the toolset management nodes. The nodes under this button are discussed next.

Create: This node is used to add a toolset to the toolbar.

Particle: This node is used to access various toolsets such as **P_DustHit**, **P_FogBox**, **P_RainBox**, **P_SnowBox**, **P_Sparks**, **P_Streaky**, **P_Trail**, **P_VolumetricLight**, and **P_Waveform** to create various particle effects.

Delete: This node is used to delete the toolset created earlier.

Other



This button is used to access the script and viewer management nodes. The nodes under this button are discussed next.

Assert: This node is used to test the validity of a user-specified TCL expression.

AudioRead: This node is used to import the audio file and view it in the **Curve Editor** panel. Then, you can line the audio file with your composition.

Backdrop: This node is used to visually group nodes in the **Node Graph** panel.

DiskCache: This node is used to cache the data stream to hard drive.

Dot: This node is used to make bends. The input data passed through this node remains unchanged.

Input: This node is used to add an input arrow to a group.

Output: This node is used to add an output arrow to a group.

NoOp: This node is used to pass data stream unchanged through itself.

PostageStamp: This node displays a very small thumbnail of the image in the **Node Graph** panel.

Group: This node is a nesting container for a group of nodes.

Precomp: This node is used to read a precomp script.

StickyNote: This node is used to add a sticky note to the **Node Graph** panel.

All plugins: This option is used to display a menu that lists all nodes and plugins including unsupported nodes, if any.

FurnaceCore



This button is used to access the Furnace plugin nodes. The nodes under this button are discussed next.

F_Align: This node is used to calculate the positions of 4 corner pins so that each frame in one shot is aligned with the corresponding frame in a second reference shot.

F_DeFlicker2: This node is used to remove flicker from an image.

F_MatchGrade: This node is used to analyze two clips and it attempts to make their color range look the same.

F_ReGrain: This node is used to add grain to a sequence.

F_RigRemoval: This node is used to remove foreground elements from an image sequence.

F_Steadiness: This node is used to calculate a four-corner pin using Global Motion Estimation (GME) on a clip so that all the frames in a clip are locked to a single frame.

F_WireRemoval: This node is used to digitally remove wire from a clip.

MANAGING PROJECT FILES

NukeX is a production-proven node based compositing software. A NukeX project consists of a network of operators called nodes. These nodes are connected together to read, process, and manipulate images and then generate the final output. NukeX project files are called scripts. In this section, you will learn about opening, saving, and closing the scripts. Also, you will learn about various project settings that need to be done in a script.

To understand the concepts explained in this chapter, you need to download *c01_nuke_7_prt.zip* file from www.cadcim.com, as the concepts have been explained using the files contained in this zipped file. The path of the file is as follows: *Textbooks > Animation and Visual Effects > NukeX > The Foundry NukeX 7 for Compositors*. Now, navigate to the *Documents* folder and create a new folder with the name *Nuke_Projects*. Extract the contents of the downloaded zip file to *\Documents\Nuke_Projects*.



Note

1. Every tutorial zip file that you download from the CADCIM website contains the folder *Media_Files*. This folder contains all media files related to the chapter.

2. Make sure that the *QuickTime* is installed on your system.

Opening Scripts

The default extension of a NukeX script file is *.nk*. To open a NukeX script, follow the steps given below:

1. Choose **File > Open** from the menu bar; the **Script to open** dialog box is displayed, as shown in Figure 1-8.

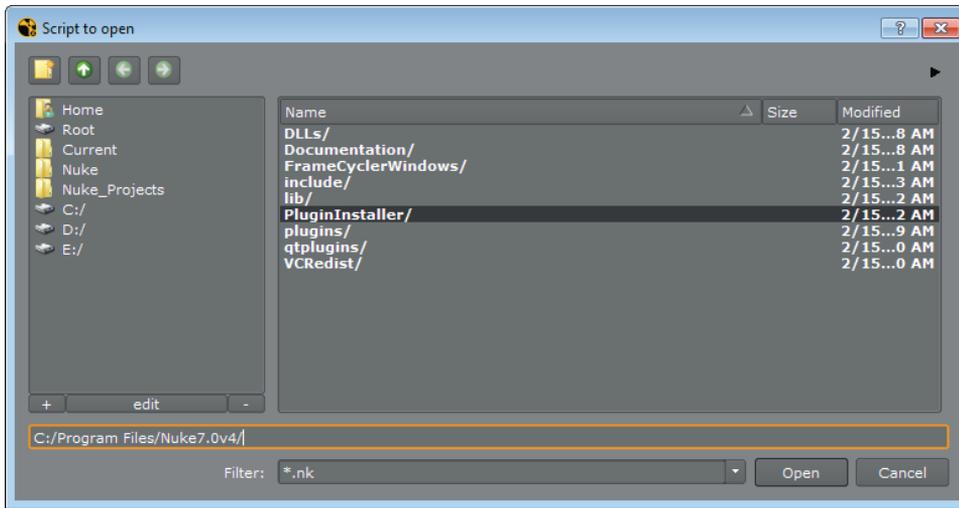


Figure 1-8 The *Script to open* dialog box

- Choose **Home** from the Navigation Controls section of the dialog box, refer to Figure 1-9. In the Directory List section, choose **Documents > Nuke_Projects > c01_prt > c01_prt_01.nk**. Next, choose the **Open** button; the selected NukeX script is displayed in the NukeX window.



Note

Now onwards, the path to the *Nuke_Projects* folder will be referred to as **Home > Documents > Nuke_Projects** throughout this textbook.

Saving Scripts

To save the changes made in the script, choose **File > Save** from the menu bar. To save and upgrade to the next version, choose **File > Save New Version** from the menu bar. The autosave feature of NukeX is used to make automatic backup of the scripts. The **autosave** and **Save New Version** options are discussed next.

Autosaving a Script

The autosave feature is a very useful option for recovering script files in case the system crashes. The default extension of an autosave script is *.autosave*. To set the autosave options, choose **Edit > Preferences** from the menu bar; the **Preferences** dialog box will be displayed, as shown in Figure 1-10. By default, the **autosave filename** field is set to [firstof [value root. name] [getenv NUKE_TEMP_DIR]/].autosave. As a result, the autosave files are saved in the same folder as the script files. If you enter 300 in the **force autosave after** field, the script is forced autosaved after 5 minutes.

The **autosave after idle for** field is used to specify the time after which the script will be autosaved, if the mouse is not moved or a key is not pressed. The default value of this field

is 5 seconds. The **force autosave after** field is used to specify the time after which the script will be force autosaved.

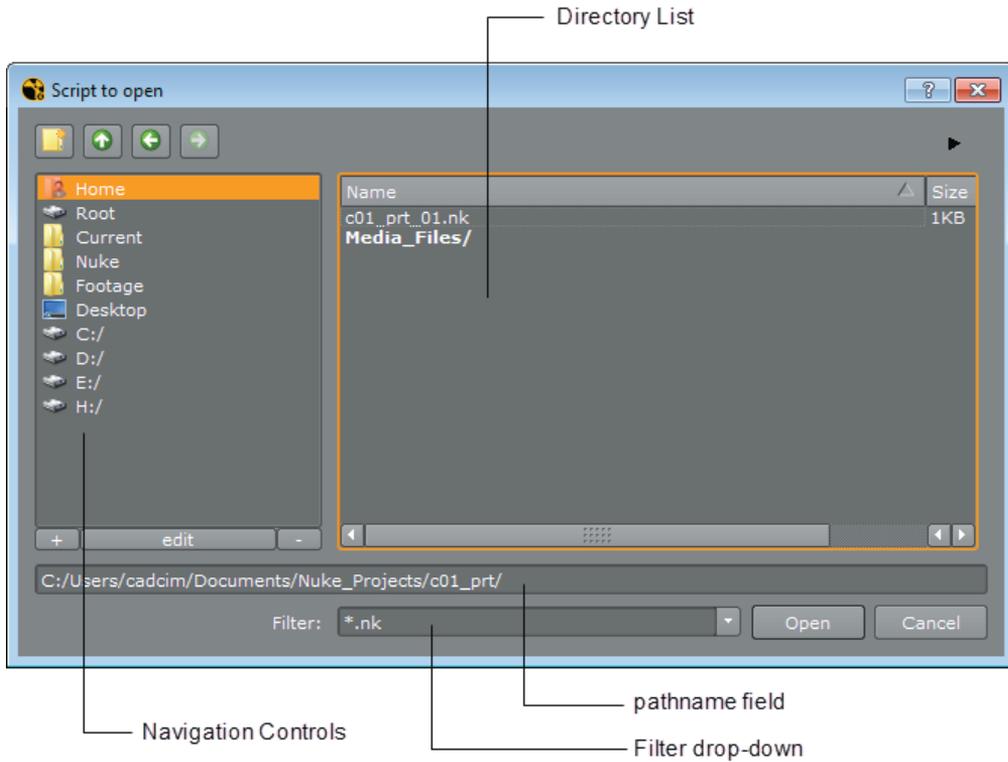


Figure 1-9 The Navigation Controls and Directory List sections in the *Script to open* dialog box

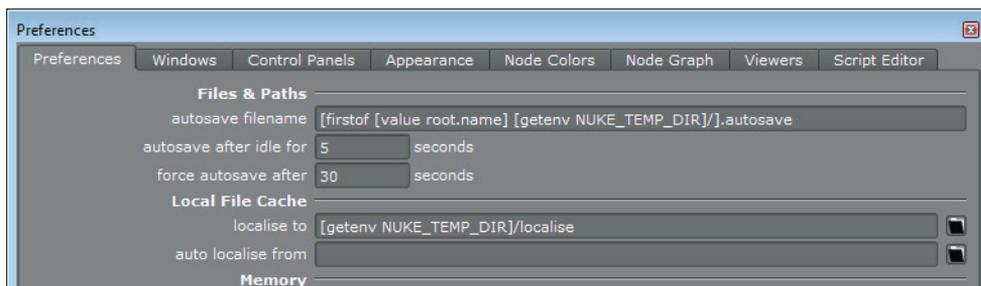


Figure 1-10 Partial view of the *Preferences* dialog box

Choose the **Save Prefs** button to save the preferences and then the Close (x) button to close the **Preferences** dialog box.

**Note**

1. The autosave function references the script file to create the backup. Therefore, you must save the script file first.
2. If you exit NukeX without saving the changes made in the script and a backup file is available for that script, the **Restore autosave file** message box will be displayed on reopening the script next time, refer to Figure 1-11.

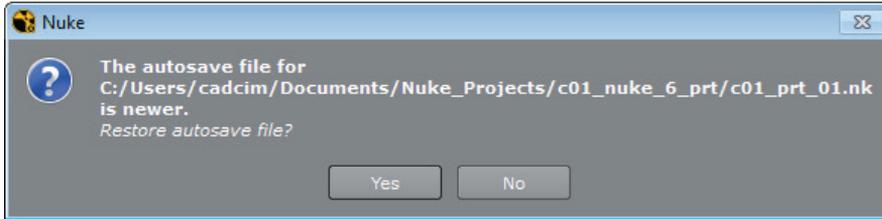


Figure 1-11 The Restore autosave file message box



Tip: To turn off the automatic backup, set the **autosave after idle for** and **force autosave after** fields to zero.

Saving New Version

You can also save different versions of your script by using the **Save New Version** option. To save the first version, save the script with the name *name_of_script_v1.nk* by choosing **File > Save As** from the menu bar. To save the next updated version of the script, choose **File > Save New Version** from the menu bar or press ALT+SHIFT+S. On doing so, the script will be saved with the name *name_of_script_v2.nk*. If you press ALT+SHIFT+S again, the file will be saved with the name *name_of_script_v3.nk*.



Tip: 1. If you are not sure about the path of the script you are working on, press **Q**; the **Current Info** message box will be displayed with the path and the name of the script displayed in it.

2. If you are not sure about the function of a shortcut key, choose **Help > Key Assignments** from the menu bar; the **Nuke key assignments** message box will be displayed. From this message box, you can check all commands and shortcuts associated with them.

Closing a Script

To close a NukeX script, choose **File > Close** from the menu bar; a message box will be displayed prompting you to save the unsaved changes, if any. In the message box, choose the **Yes** button to save the changes made; the **No** button to discard the changes; and the **Cancel** button to exit it. NukeX is relaunched, if you choose the **Yes** or **No** button from the message box.

**Note**

Choosing **File > Close** from the menu bar or pressing CTRL+W is a quick way to clear the memory used by the NukeX script. This command reloads the *init.py* and *menu.py* files which are used to configure NukeX.

SPECIFYING PROJECT SETTINGS

Before you start working on a new script, you need to specify the project settings. To specify the project settings, you need to follow the steps given below:

1. Choose **File > Open** from the menu bar; the **Script to open** dialog box will be displayed. Next, choose **Home > Documents > Nuke_Projects > c01_prt > c01_prt_01.nk** from the dialog box. Choose the **Open** button from the **Script to open** dialog box.
2. Choose **Edit > Project Settings** from the menu bar or place the cursor over the **Node Graph/Curve Editor/Dope Sheet** pane and press S; the **Project Settings** properties panel will be displayed with the **Root** tab chosen, as shown in Figure 1-12.

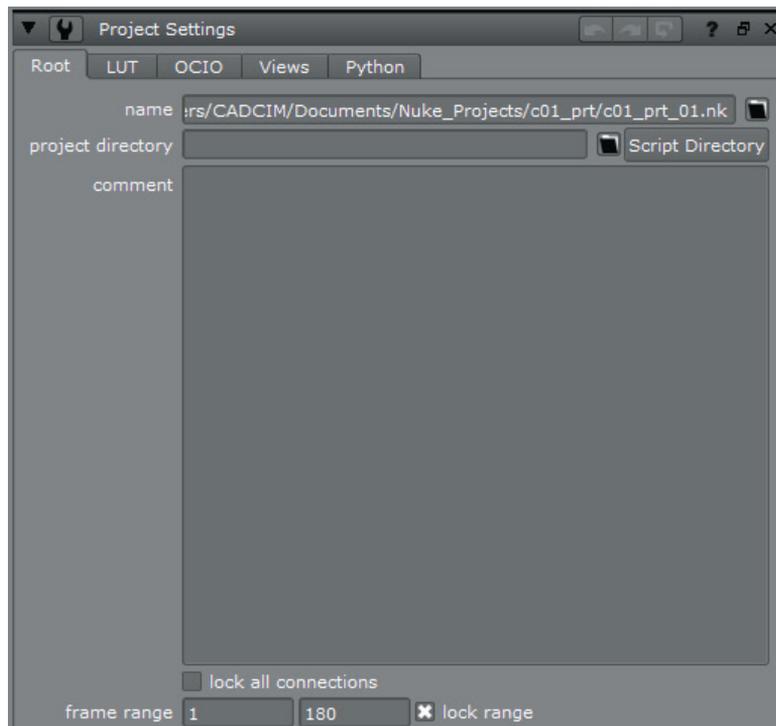


Figure 1-12 Partial view of the Project Settings properties panel



Tip: If the mouse pointer is on the **Viewer** pane and you press S, the **Viewer#** properties panel will be displayed instead of the **Project Settings** panel.

3. In the **Root** tab, the name of the script file is already displayed in the **name** field. Click on the folder icon located on the right of the **project directory** field; the **root:Select file(s)** dialog box will be displayed, as shown in Figure 1-13. In this dialog box, choose **Home > Documents > Nuke_Projects > c01_prt** and then choose the **Open** button to define the selected directory as the project directory, refer to Figure 1-13.

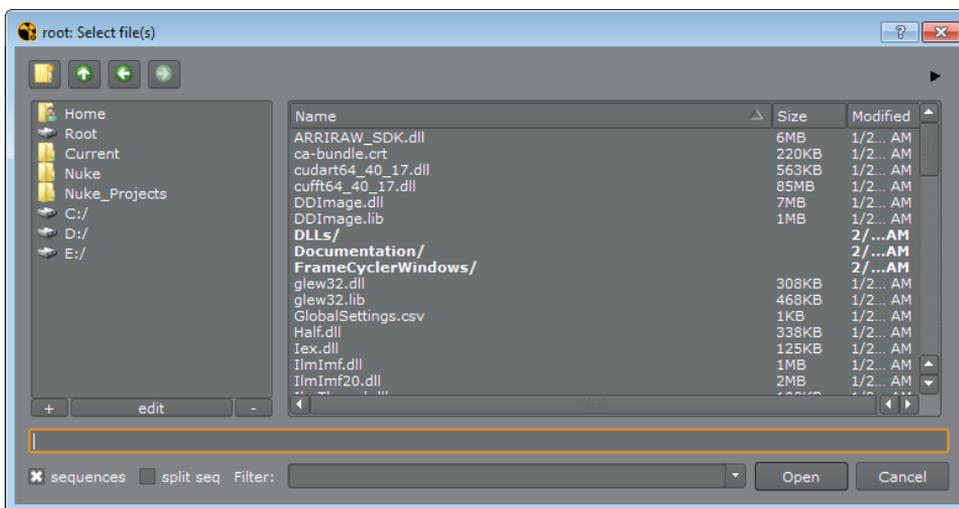


Figure 1-13 The root: Select file(s) dialog box

- To set the duration of the project, enter the required values in the two fields corresponding to the **frame range** parameter in the **Project Settings** properties panel.

The resolution of the footage used in this script is 1280 x 720 pixels (HD format). Therefore, 24 is displayed in the **fps** field, as it is the default frame rate for the HD format.

- Select the **1280x720** option from the **full size format** drop-down, as shown in Figure 1-14. Next, choose **File > Save** from the menu bar to save the script.

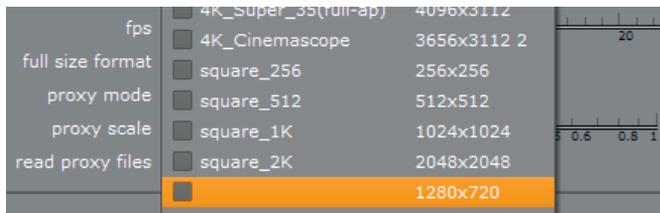


Figure 1-14 Partial view of the full size format drop-down

The options in the **full size format** drop-down are used to set the format of the project. By default, the 2K_Super_35(full-ap) 2048x1556 option is selected in this drop-down.

LOADING IMAGE SEQUENCES

To start compositing, you need to import still images or image sequences for the background and foreground elements. To import an image sequence, follow the steps given next:

- Choose **File > New** from the menu bar; a new blank script file will be displayed in the NukeX interface.

2. Choose the **Image** button from the **Nodes** toolbar; the **Image** menu will be displayed. Next, choose **Read** from the menu, refer to Figure 1-15; the **Read File(s)** dialog box will be displayed, as shown in Figure 1-16.
3. In this dialog box, choose **Home** from the Navigation Controls section of the dialog box. Next, choose **Documents > Nuke_Projects > c01_prt > Media_Files > 100_4172.MOV** from Directory List section and then choose the **Open** button; the **Read1** node will be inserted in the **Node Graph** panel, as shown in Figure 1-17.
4. Select the **Read1** node in the **Node Graph** panel and then press 1; the selected footage will now be displayed in the **Viewer1** panel, as shown in Figure 1-18. You will notice that the **Read1** node is connected to the **Viewer1** node in the **Node Graph** panel, refer to Figure 1-19.

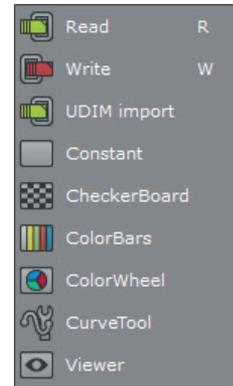


Figure 1-15 The Image menu

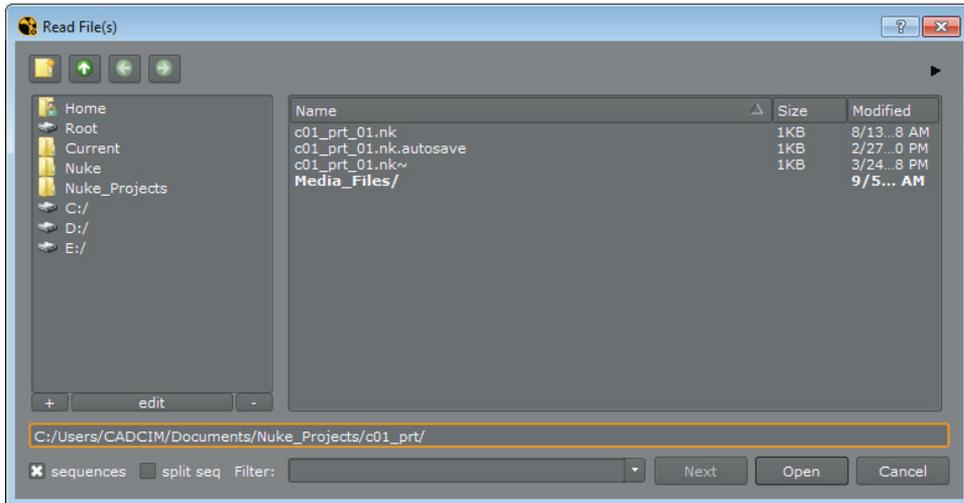


Figure 1-16 The Read File(s) dialog box

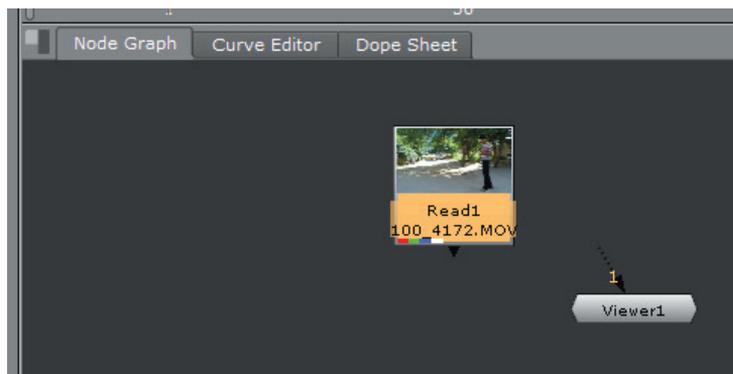


Figure 1-17 The Read1 node inserted in the Node Graph panel



Figure 1-18 The footage displayed in the **Viewer1** panel

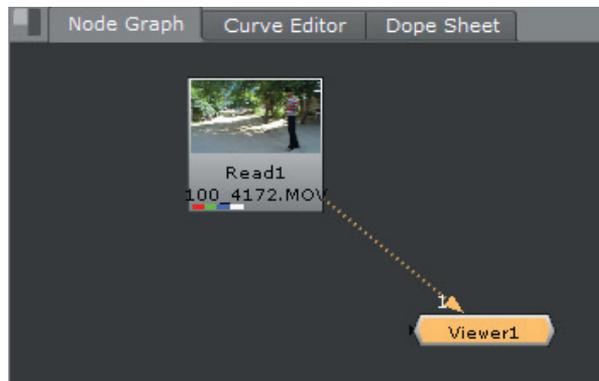


Figure 1-19 The **Read1** node connected to the **Viewer1** node



Note

1. To open multiple files at a time, select the desired files using the CTRL key and then choose the **Open** button from the **Read File(s)** dialog box.
2. By default, NukeX adds alpha channel to an image sequence, if the sequence has RGB channels but no alpha channel.
3. NukeX converts the native colorspace format of an input image into linear colorspace which is the native colorspace format of NukeX. However, you can select the required colorspace from the **colorspace** drop-down of the **Read** node properties panel.
5. Select the **Read1** node in the **Node Graph** panel and press DEL to delete it from the **Node Graph** panel.

Next, you will import a *TIFF* sequence into the script.

6. Open the Windows Explorer and browse to `\Documents\Nuke_Projects\c01_prt\Media_Files\Cube_Occlusion`; the image sequence will be displayed in the Windows Explorer, as shown in Figure 1-20. You will notice that in the Window Explorer, *TIFF* files from 1 to 89 are displayed.

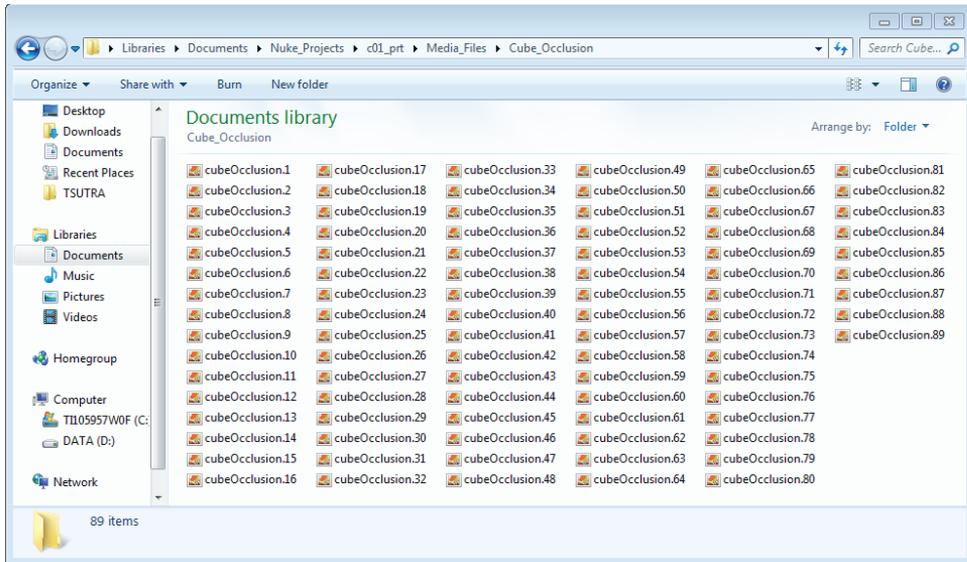


Figure 1-20 The image sequence displayed in the Windows Explorer

7. Insert a **Read** node to the **Node Graph** panel, as described in Step 2. In the **Read file(s)** dialog box, choose **Home > Documents > Nuke_Projects > c01_prt > Media_Files > Cube_Occlusion**.

You will notice that the **sequences** check box is selected at the bottom left corner of the dialog box. As a result, NukeX gathers the images and displays them as a group, *cubeOcclusion.#.tif 1-89*, as shown in Figure 1-21.

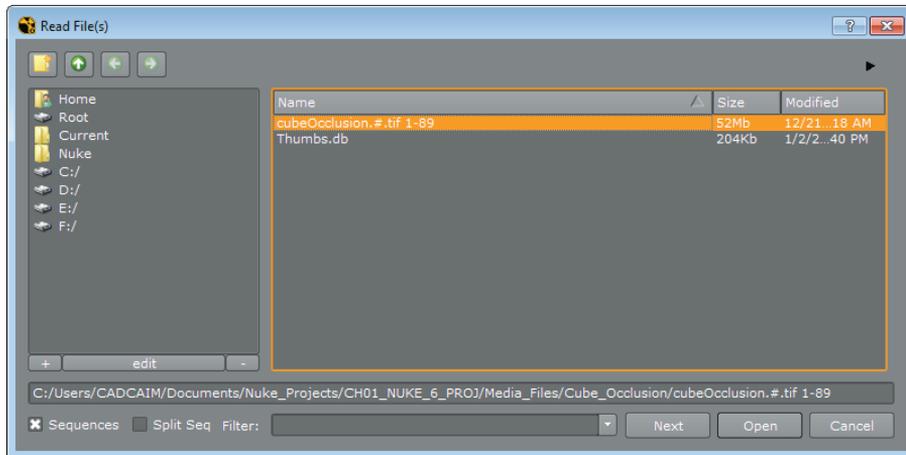


Figure 1-21 The gathered sequence displayed in the Read File(s) dialog box

8. Select **cubeOcclusion.#.tif 1-89** in the Directory List section of the dialog box and then choose the **Open** button to import the sequence. Notice in the **Read1** node properties panel, the default frame range 1-89 will be displayed, refer to Figure 1-22.

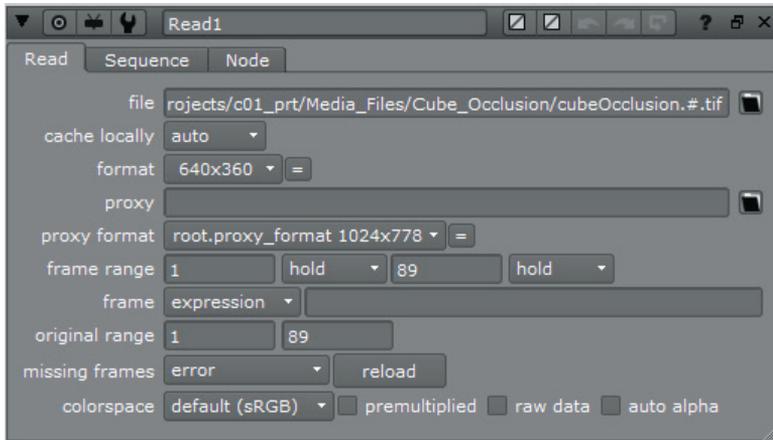


Figure 1-22 The *Read1* node properties panel displaying the frame range



Note

1. You will notice a symbol # in the name of the gathered sequence. It is called as frame number variable. It indicates that number is in 1-digit increment format. For a 4-digit increment format, such as *cubeOcclusion0001.tif*, the frame number variable would be #####.

2. To load a single image from the Windows Explorer, drag and drop the image into the **Node Graph** panel, refer to Figure 1-23. To import an image sequence, drag and drop the folder that contains the image sequence into the **Node Graph** panel, refer to Figure 1-24.

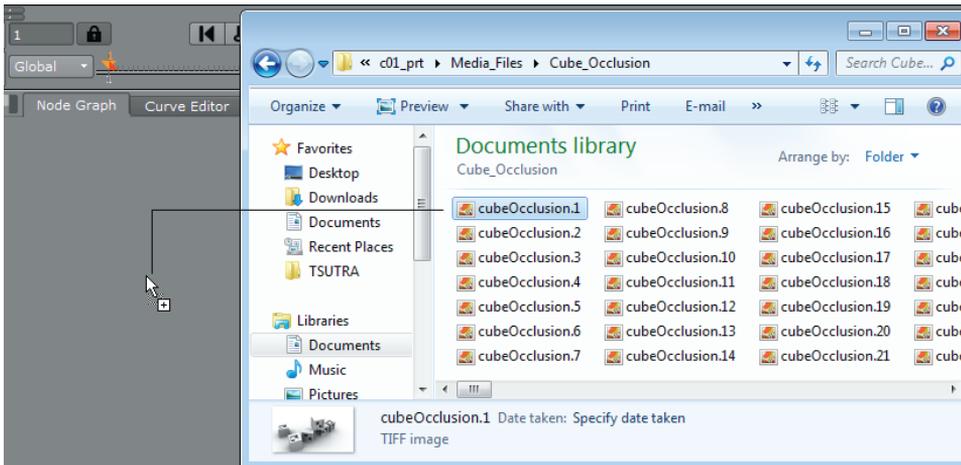


Figure 1-23 Dragging an image from the Windows Explorer

WORKING WITH PROXIES

The Proxy modes are used for the proxy scaling of a footage to speed up the rendering process and the display calculations. You can specify global proxy settings in the **Project Settings** panel. You can also enable the proxy mode for an individual image sequence by using the **Read** node. To set up proxy formats, you need to follow the steps given next:

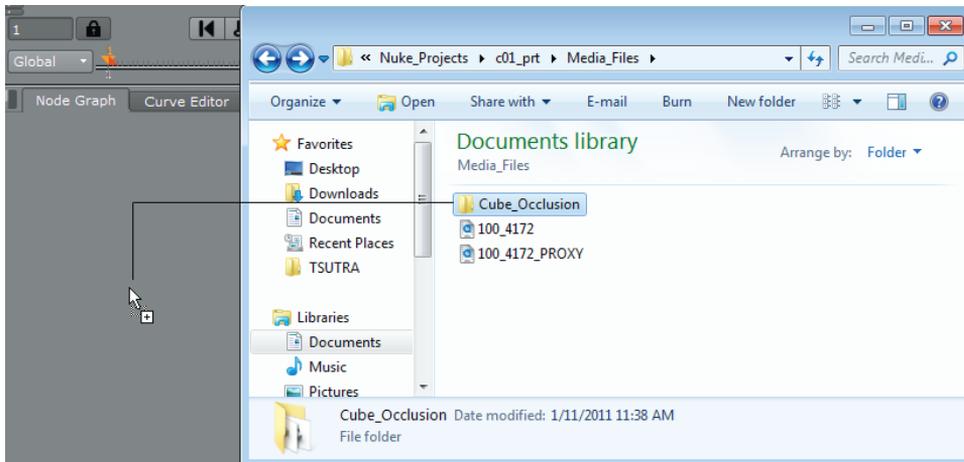


Figure 1-24 Dragging a folder from the Windows Explorer

1. Choose **File > Open** from the menu bar; the **Script to open** dialog box will be displayed. Next, choose **Home** from the Navigational Controls section of the dialog box. In the Directory List section, choose **Documents > Nuke_Projects > c01_prt > c01_prt_01.nk**. Next, choose the **Open** button; the selected NukeX script will be displayed in the NukeX window.
2. Choose **Edit > Project Settings** from the menu bar; the **Project Settings** panel will be displayed in the **Properties Bin** with the **Root** tab chosen. Select the **proxy mode** check box in this panel.
3. Select **1280x720** from the **full size format** drop-down.
4. Select **format** from the **proxy mode** drop-down.
5. Select **new** from the **proxy format** drop-down; the **New format** dialog box will be displayed. Type **HDNew** in the **name** field. Set the values **640** and **360** in the **file size w** (width) and **file size h** (height) fields, respectively, refer to Figure 1-25. Next, choose the **OK** button; a new format will be created.

You will notice in the **Viewer1** panel that **HDNew** is displayed as the new format. Also, the **toggle proxy mode** button is activated, as shown in Figure 1-26.



Tip: Press **CTRL+P** on your keyboard to toggle proxy mode.

6. Select **scale** from the **proxy mode** drop-down; the **proxy scale** field will be displayed. Next, set the value of the **proxy scale** field to specify the factor by which you want to scale the dimensions of the images.

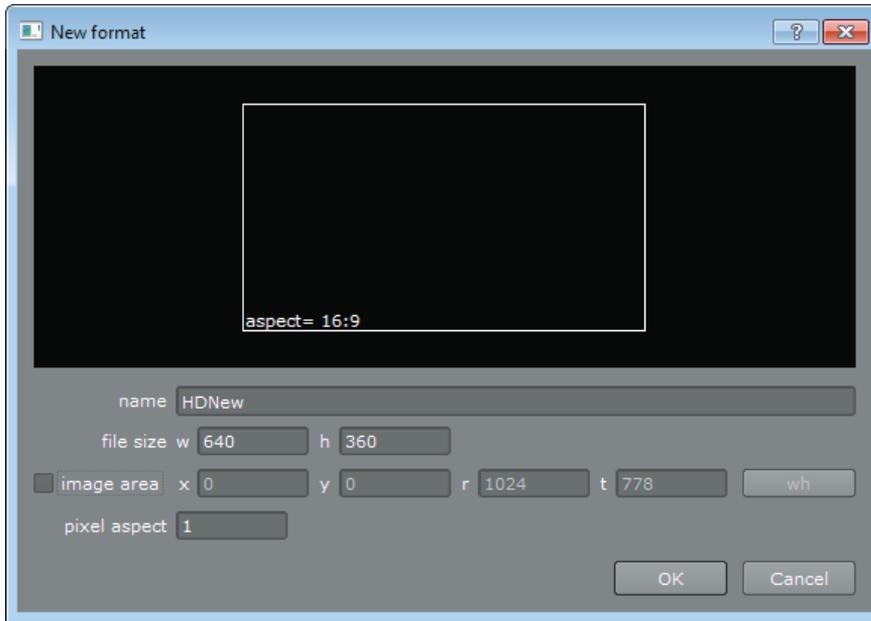


Figure 1-25 The New Format dialog box



Figure 1-26 The toggle proxy mode button activated

Proxy Settings in the Read Node

Proxies can be specified in the **Read** node properties panel by using the **proxy** field. To set the proxy options in the **Read** node, you need to follow the steps given next:

1. Choose **File > New** from the menu bar to create a new script. Next, choose the **Image** button on the **Nodes** toolbar; the **Image** menu will be displayed. Choose **Read** from the menu or press R; the **Read File(s)** dialog box will be displayed. In this dialog box, choose

Home > Documents > Nuke_Projects > c01_prt > Media_Files > 100_4172.MOV and then choose the **Open** button; the **Read1** node is inserted in the **Node Graph** panel.

2. Select the **Read1** node in the **Node Graph** panel and press 1 to connect the **Read1** node to the **Viewer1** node; the output of the **Read1** node will be displayed in the **Viewer1** panel.
3. In the **Read1** node properties panel, click on the folder icon located on the right of the **proxy** field; the **Read1: Select file(s)** dialog box will be displayed. In this dialog box, choose **Home > Documents > Nuke_Projects > c01_prt > Media_Files > 100_4172_PROXY.mov** and then choose the **Open** button; NukeX will use the proxy file to speed up the rendering process and display calculations.

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 combination of shortcut keys is used to cycle though the tabs in a pane?
 - (a) CTRL+T
 - (b) CTRL+S
 - (c) ALT+P
 - (d) CTRL+P
2. Which of the following combination of shortcut keys is used to invoke the **Preferences** dialog box?
 - (a) SHIFT+P
 - (b) SHIFT+S
 - (c) SHIFT+J
 - (d) CTRL+S
3. NukeX is a _____ based compositing software application.
4. The **Read** node is used to _____ image sequences into a script.
5. In NukeX, you can save and restore up to _____ layouts.
6. The saved project files in NukeX are called _____ files.
7. The _____ shortcut key is used to insert the **Read** node in the **Node Graph** panel.
8. The W shortcut key is used to insert the **Write** node in the **Node Graph** panel. (T/F)
9. By default, NukeX adds alpha channel to an image. (T/F)
10. NukeX converts all imported sequences to its native linear colorspace. (T/F)

Review Questions

Answer the following questions:

- Which of the following shortcut keys is used to open the **Project Setting** panel?
 - J
 - L
 - S
 - P
- Which of the following combination of shortcut keys is used to toggle the proxy mode?
 - CTRL+P
 - CTRL+S
 - CTRL+K
 - CTRL+M
- Which of the following combination of shortcut keys is used to save and upgrade to the next version of the script?
 - ALT+SHIFT+S
 - ALT+SHIFT+J
 - ALT+SHIFT+U
 - ALT+SHIFT+K
- The **Grain** node is used to add _____ grain to a footage.
- _____ are used to speed up rendering and display calculations.
- The _____ and _____ files are used to configure NukeX.
- The _____ node is used to make precision adjustments to the levels of saturation in a range of hues.
- The Node Graph is also known as _____.
- The _____ panel is used to build a network of nodes.
- The _____ option from the Content menu is used to split a pane horizontally.
- You need to press and hold CTRL and then click on a node tile to display its properties panel in a floating window. (T/F)
- You can load image sequences into NukeX by using the Windows Explorer. (T/F)

Answers to Self-Evaluation Test

1. a, 2. b, 3. node, 4. load, 5. six, 6. script, 7. R, 8. T, 9. T, 10. T