VAPOR User's Guide for WRF Typhoon Research

Version 1.5

Aug, 2009

Web Links updated April 2013

Minsu Joh

(msjoh@kisti.re.kr)

Supercomputing Center

Korea Institute of Sci.&Tech. Information

Contents

Chapter 1: Overview

1.1 Introduction

VAPOR is the Visualization and Analysis Platform for atmospheric, Oceanic, and solar Research. VAPOR was developed at CISL / NCAR (Computational and Information System Laboratory / National Center for Atmospheric Research) to provide interactive visualization and analysis of numerically simulated results in fluid dynamics.

The WRF (Weather Research and Forecasting) model is a mesoscale numerical weather prediction system developed at NCAR's ESSL (Earth and Sun Systems Laboratory). It is designed to serve both operational forecasting and atmospheric research needs. It features multiple dynamical cores, a 3-dimensional variational (3DVAR) data assimilation system, and a software architecture allowing for computational parallelism and system extensibility. The WRF model is suitable for a broad spectrum of applications across scales ranging from meters to thousands of kilometers.

VAPOR supports various visualizations of the WRF model's simulation output data through a direct data conversion process.

This VAPOR user's guide is intended especially to assist the typhoon research scientists who are using the WRF model in their research. Most of the descriptions in this manual are taken from the existing VAPOR documents, but all figures are prepared independently using VAPOR (version 1.5) and WRF typhoon simulation data.

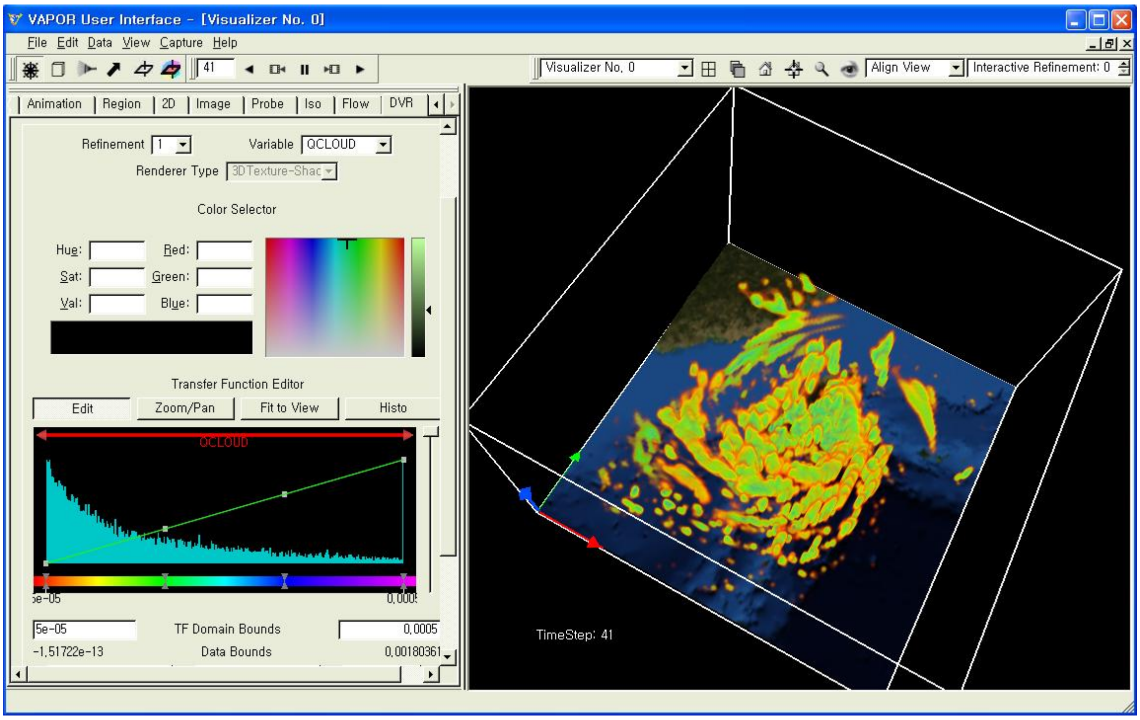
The typhoon simulation data resulted from the WRF model simulation was provided by Dr. Bill Kuo and Dr. Wei Wang at NCAR. Note that any scientific conclusions obtained by visualizing this data should be confirmed by Dr. Bill Kuo and Dr. Wei Wang.

1.2 Basic capabilities of VAPOR with WRF-ARW data

The basic capabilities of VAPOR are illustrated below using data from Typhoon Jangmi. Sections 4 and 5 of this report show how these visualizations can be created in VAPOR.

- 3D Data Rendering - Direct Volume Rendering (DVR)

Any 3D variable in the WRF data can be viewed as a density. Users can control transparency and color to view temperature, water vapor, cloud, rain, etc in 3D.
(Using the "DVR" tab:)

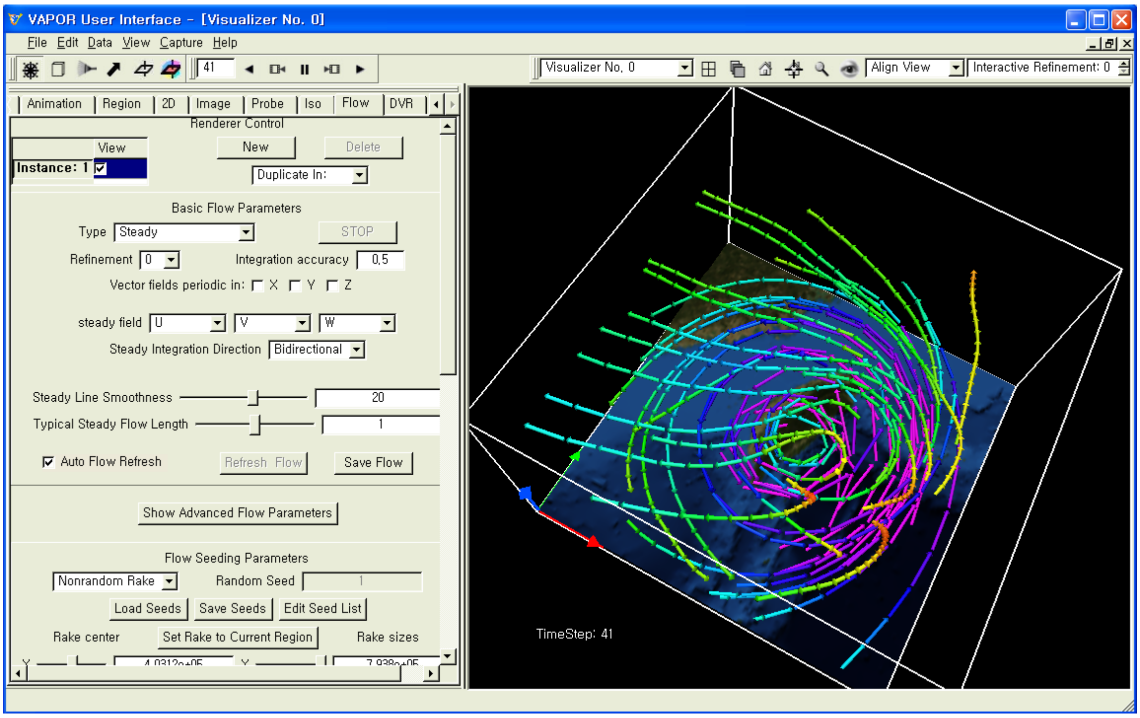


<Figure 1.1> Direct Volume Rendering of QCLOUD

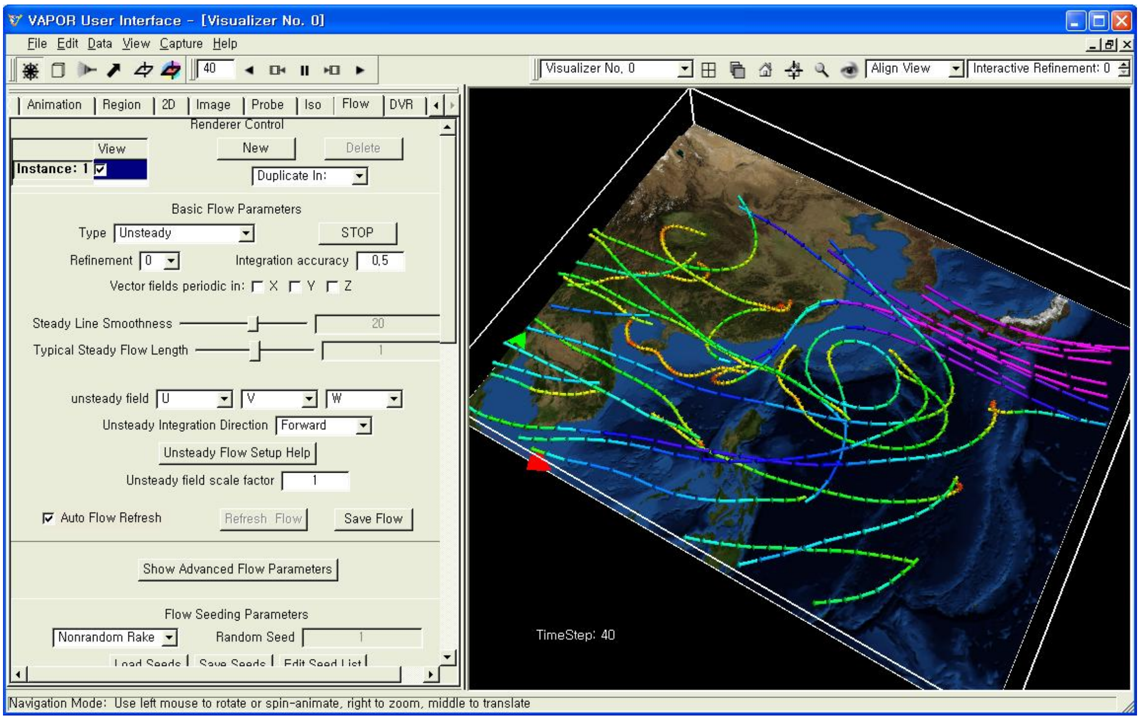
- Flow Visualization
  - Steady flow (Streamlines) visualization:

  Users can draw 2D and 3D streamlines and flow arrows, showing the wind direction and magnitude.
  - Unsteady flow (Trajectories) visualization:

  Users can draw trajectories that particles follow over time. Users can control when and where the particles are released. (Using the "Flow" tab:)



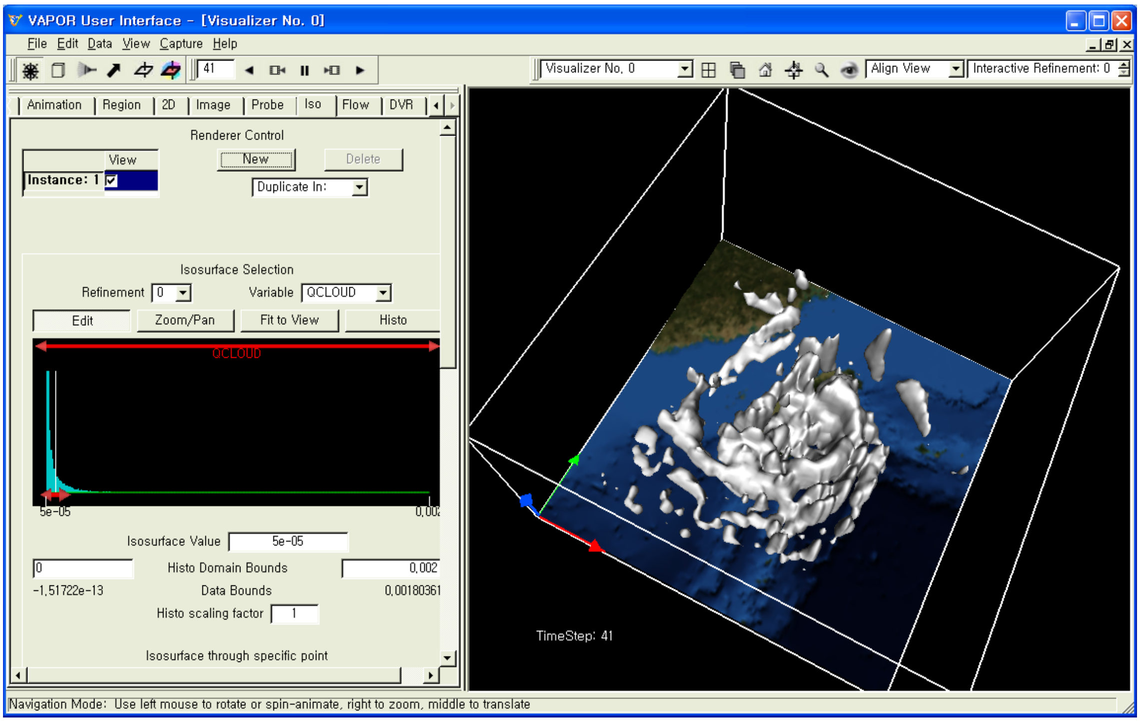
<Figure 1.2> Steady, bidirectional, nonrandom rake Flow



<Figure 1.3> Unsteady, forward, nonrandom rake Flow

· Isosurface Visualization

Any 3D variable in the WRF data can be viewed using surfaces associated with iso-values. Users can control iso-values, color and transparency of the isosurfaces. The other 3D variables in the dataset can be color-mapped onto the isosurface of the selected variable. (Using the "Iso" tab:)



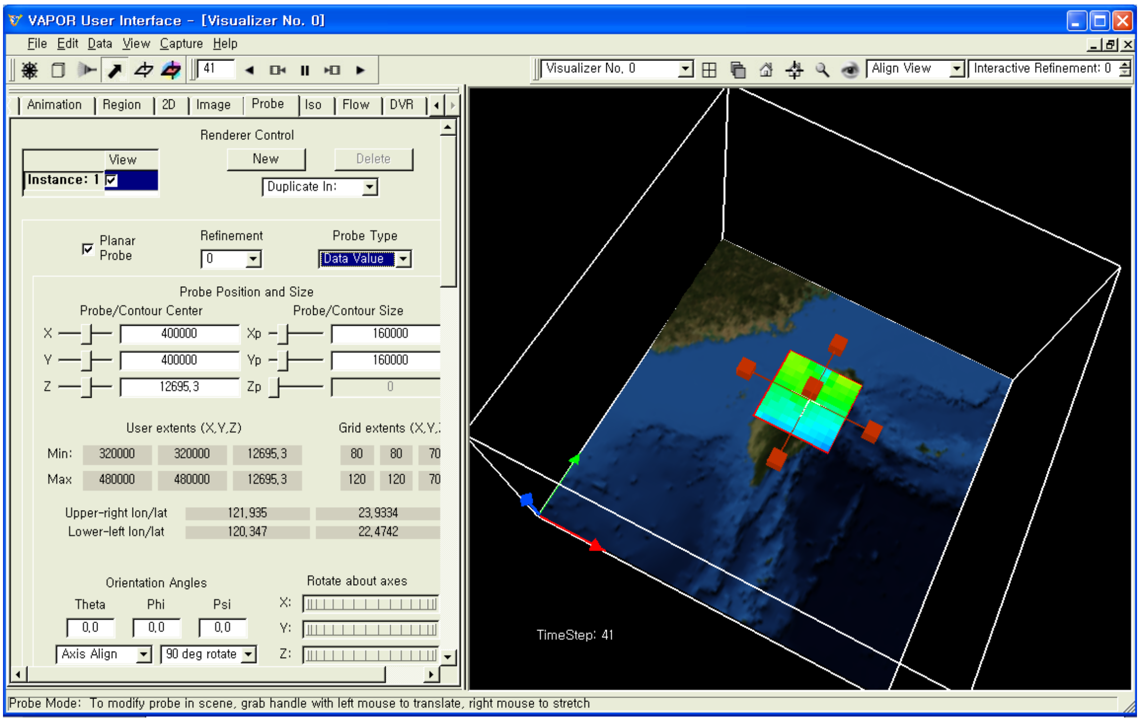
<Figure 1.4> Isosurface (value=0.00005) of QCLOUD

· Probe Planes Insertion

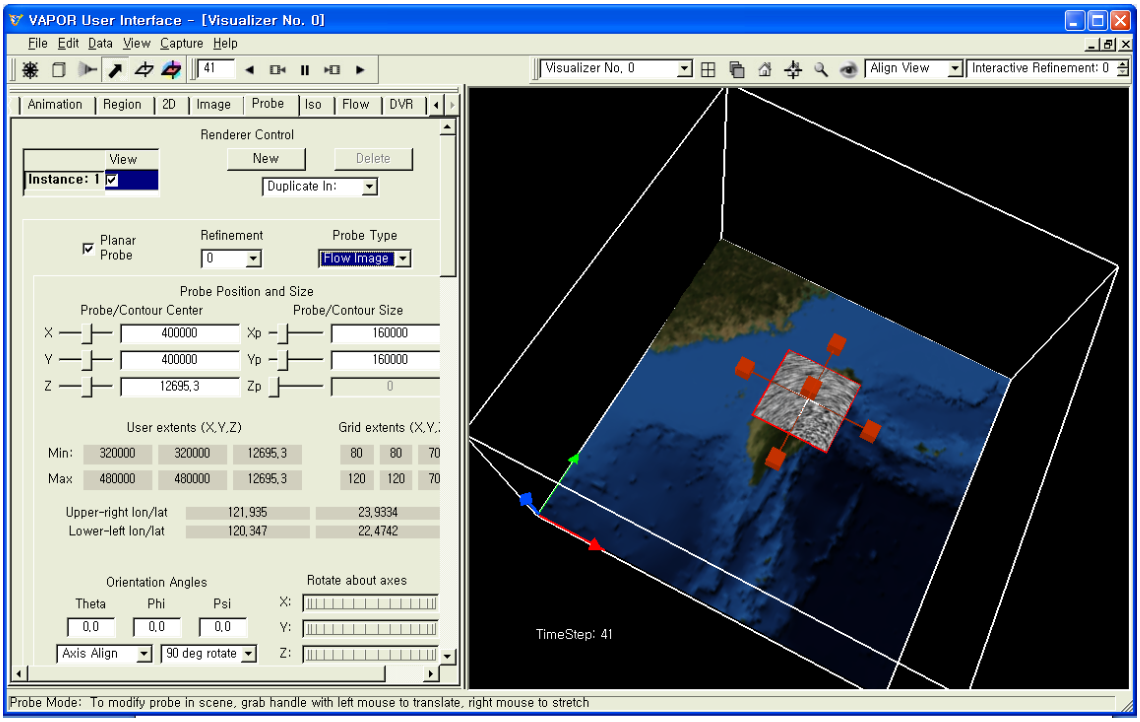
Any 3D variable in the WRF data can be intersected with arbitrarily oriented planes. Probe planes can be interactively positioned. Users can interactively pinpoint the values of a variable and establish seed points for flow integration.

Image-based flow can also be visualized in the probe plane. Image-based flow provides a means of viewing the flow as random particles moving in time.

(Using the "Probe" tab:)



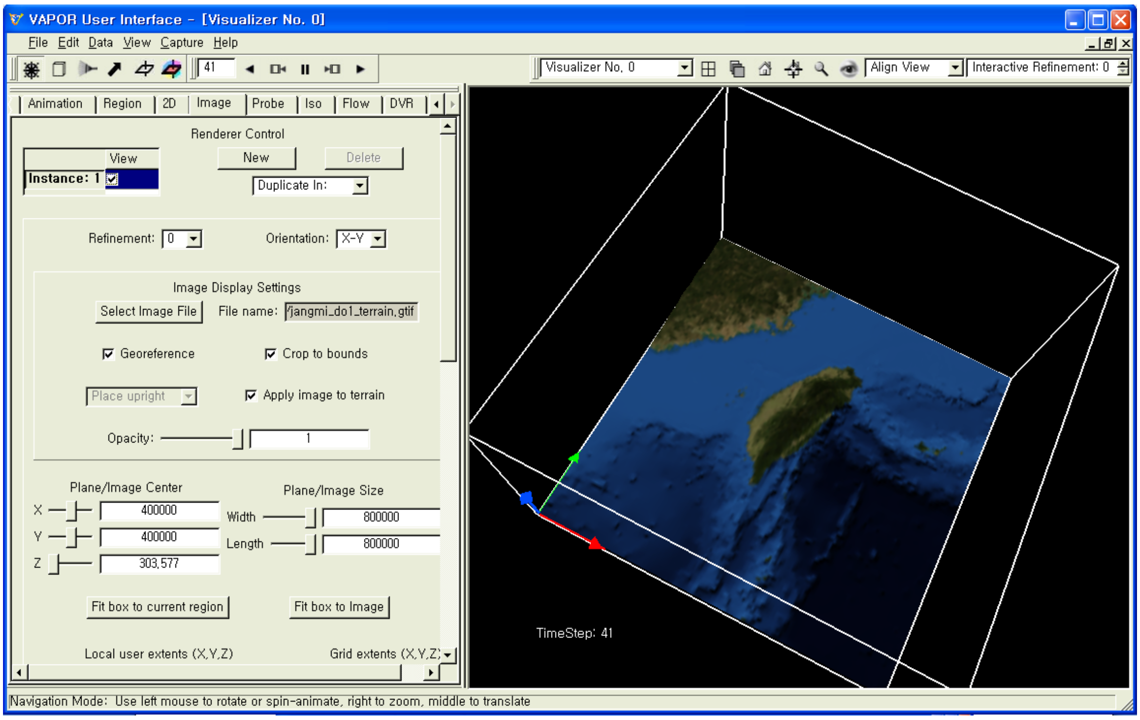
<Figure 1.5> Data-value-typed planar Probe of U



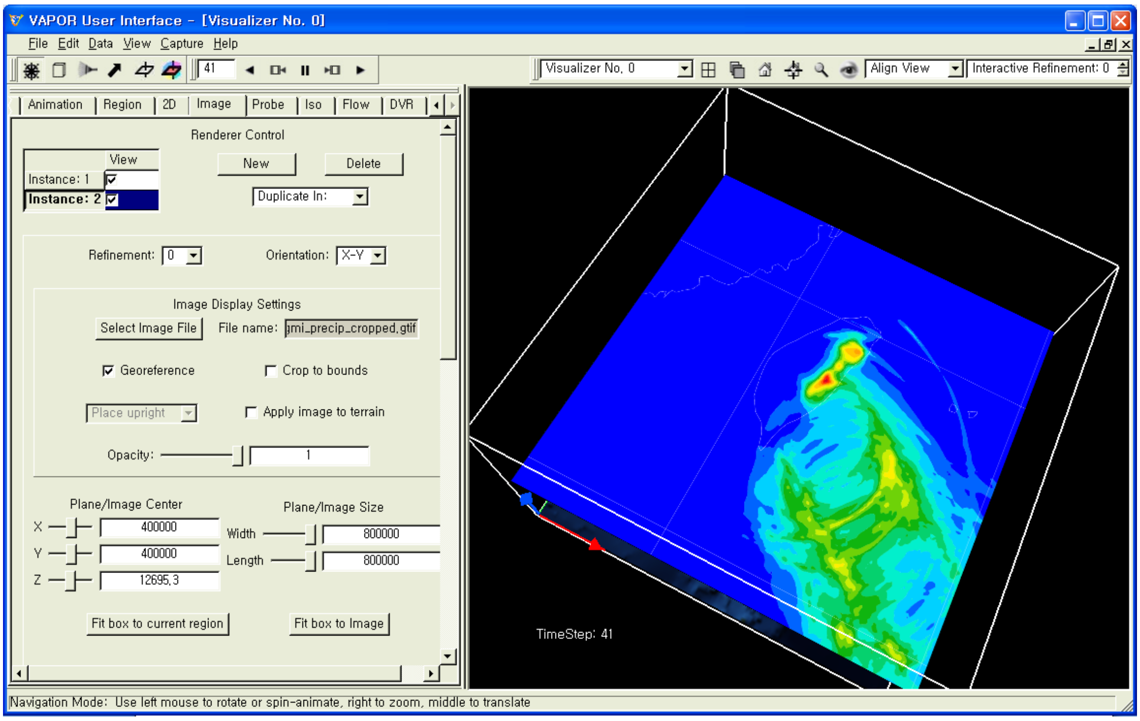
<Figure 1.6> Flow-image-typed planar Probe of U, V, W

· Image File Upload

Users can add a satellite image to display terrain. Also, users can use data plots created by NCL. (Using the "Image" tab:)



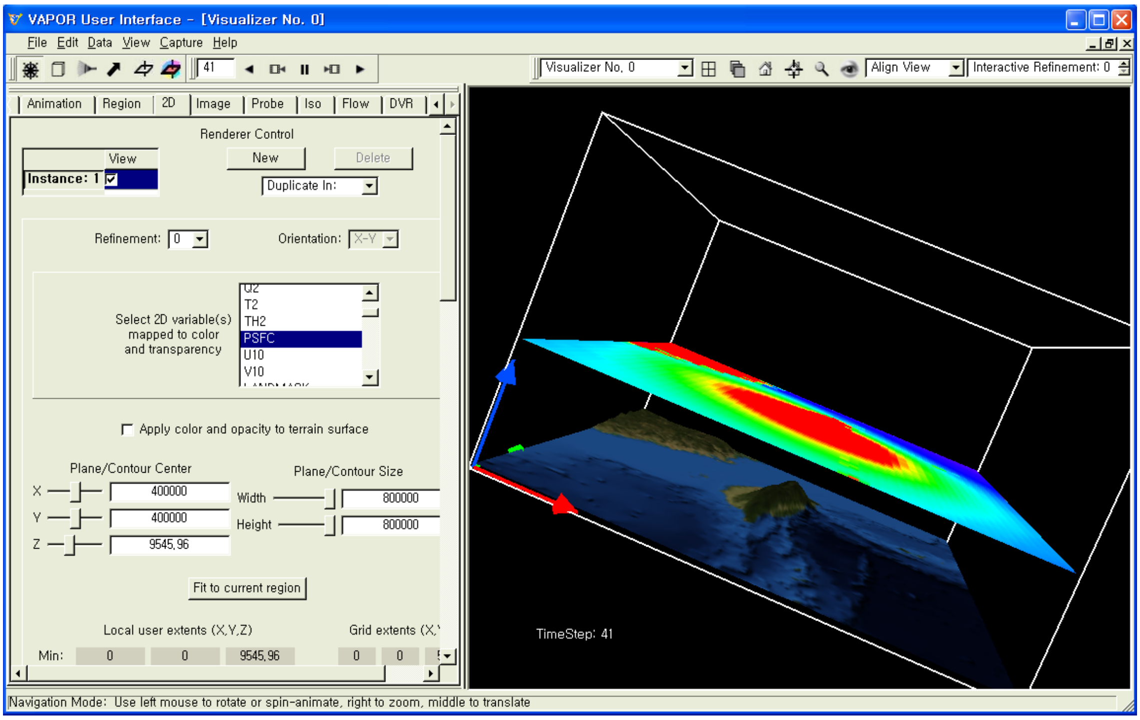
<Figure 1.7> Uploading a satellite image for representing terrain



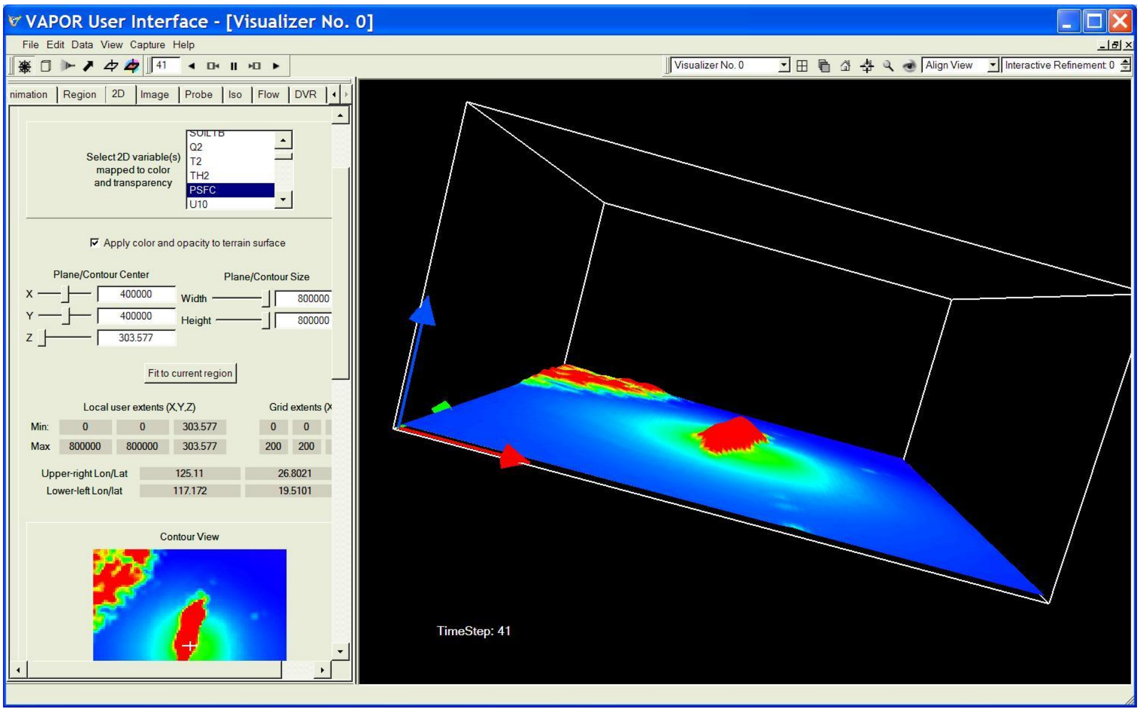
<Figure 1.8> Uploading a precipitation plot created by NCL

· 2D Data Rendering

Any 2D variable in the WRF data can be displayed in the 3D scene, either as a horizontal plane, or mapped to the terrain. (Using the "2D" tab:)



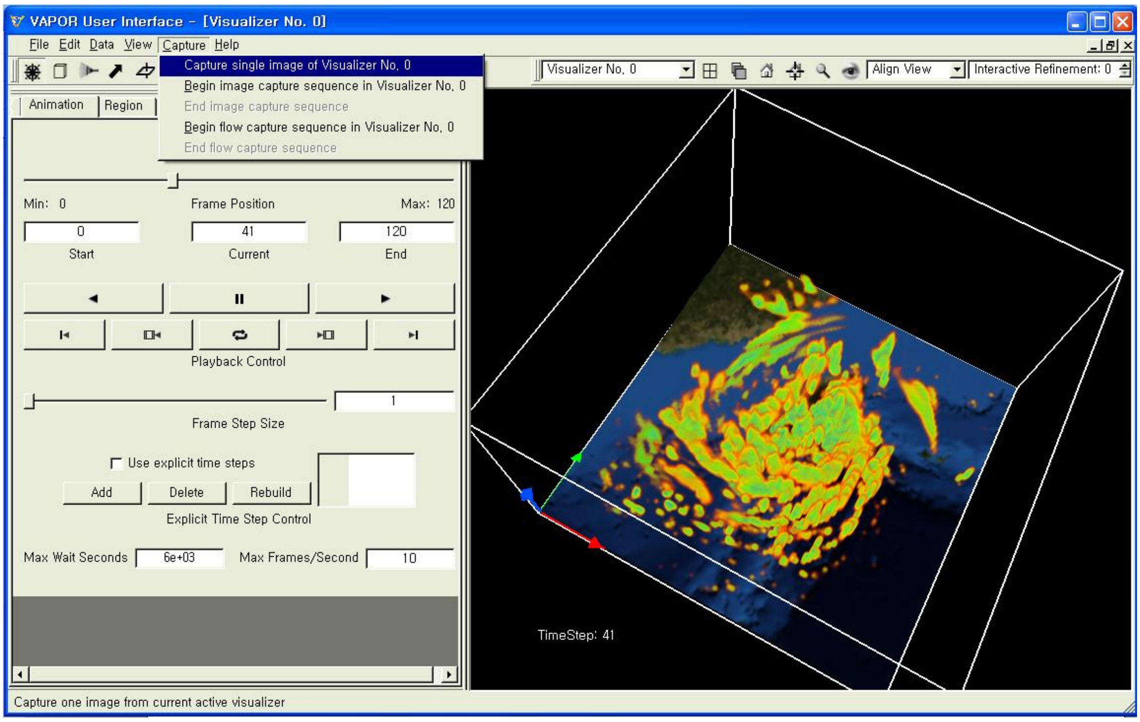
<Figure 1.9> 2D rendering of PSFC displayed in a horizontal plane



<Figure 1.10> 2D rendering of PSFC mapped to terrain surface

· Animation and Image Capture

Users can control the time-stepping of the data for interactive replaying and for recording animated sequences. Animations can be captured to single JPEG image or multiple JPEG images. (Using the "Animation" tab:)



<Figure 1.11> Capturing a single image

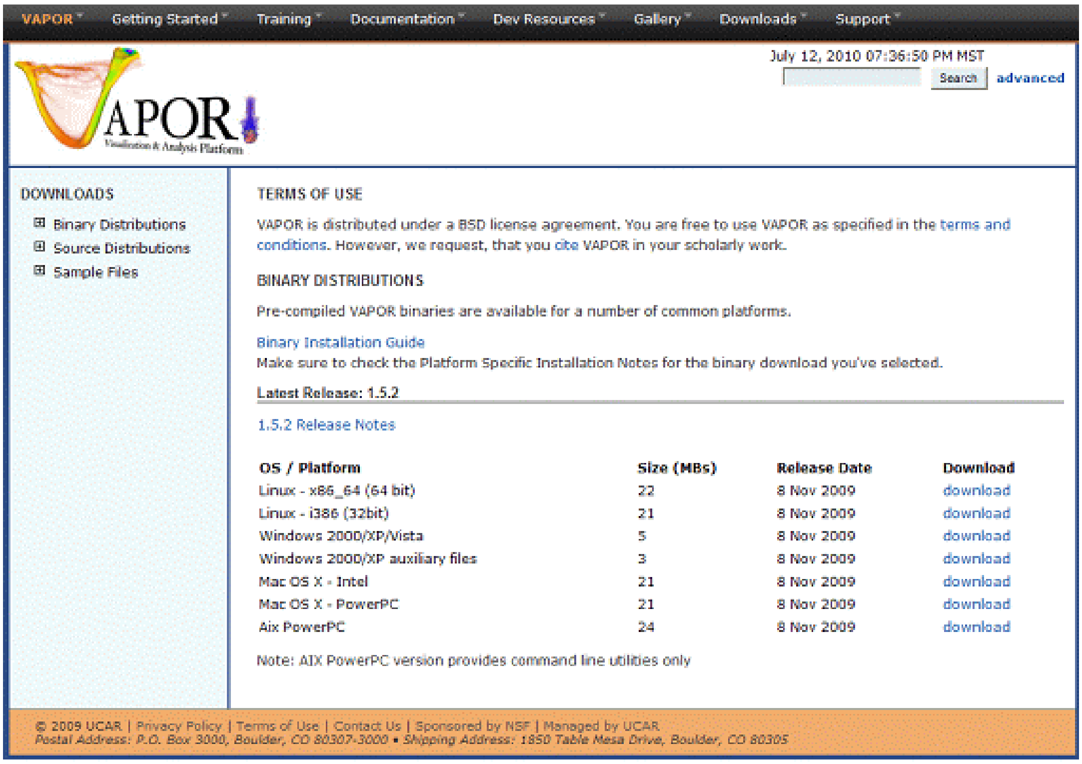
Chapter 2: VAPOR Preparation

2.1 Check Graphics Card

VAPOR is supported on Linux, Macintosh, and Windows. VAPOR works best with a recent graphics card. The advanced features of VAPOR perform best with nVidia or ATI graphics accelerators. To be sure your graphics card will work properly, it is useful to install the latest drivers, which you can obtain from the website of the card manufacturer.

2.2 Download VAPOR Installer

VAPOR installers for Linux, Macintosh, and Windows are available on the VAPOR download page, https://www.vapor.ucar.edu/page/vapor-download#Binary. For most users, a binary installation is fine. Pre-compiled VAPOR binaries are available for a number of platforms.



2.3 Installing VAPOR on Windows System

Installation instructions from pre-compiled binaries are provided at the top of the VAPOR documentation page (www.vapor.ucar.edu/doc).

First, you may need to uninstall a previously installed version of VAPOR. From the Windows Control Panel, select “Add or Remove Programs”, then select “VAPOR” and click “Remove”.

Then, you should install the newly released version of VAPOR as follows:
If you are using antivirus software, you should disable it during the installation, as it may interfere with the execution of some scripts that are run during the install.
Download the file *vapor-x.x.x-win32.msi* where ‘*x.x.x*’ is the version number. Double-click on the downloaded **.msi* file to begin the installation. The default installation will install VAPOR in the Program Files directory.

On some older versions of Windows your system may not be able to execute the msi installer. On these systems, you should in addition download the file *vapor-x.x.x-win32extras.zip*, unzipping it into the same directory where you downloaded the file *vapor-x.x.x-win32.msi*. Then double-click on Setup.exe in that directory.

Chapter 3: WRF Data Preparation

3.1 Converting a WRF data file to a VDC

A VAPOR dataset consists of two files: (1) a metadata file (.vdf) that describes an entire VAPOR data collection (VDC), and (2) a directory of multi-resolution data files where the actual data is stored.

In order to use VAPOR to visualize WRF model output data, it is necessary to convert the WRF model output data files into a VDC. VAPOR provides two command-line utilities (*wrfvdfcreate, wrf2vdf*) for this process.

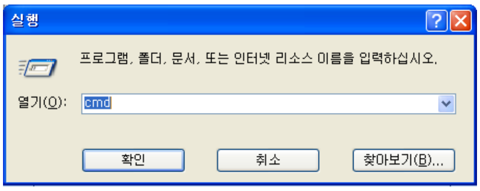
The conversion process consists of two steps: (1) creating a metadata file that describes the dataset (*wrfvdfcreate*), and (2) performing the actual data conversion (*wrf2vdf*).

Note that the WRF model output data files to be converted must be in netCDF format and have the following dimensions: WEST_EAST, SOUTH_NORTH, BOTTOM_TOP, their staggered versions and Time. The PH, PHB, and Times variables must also be present.

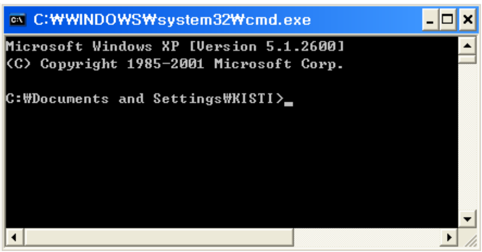
The simplest way to create a VDC on a Windows System is as follows:

- First, open a command (DOS) window.

"Windows Start" --> "Run" --> "cmd"



(a) execution dialog



(b) command window

<Figure 3.1> Open the command window

▪ Second, issue the command *wrfvdfcreate*.

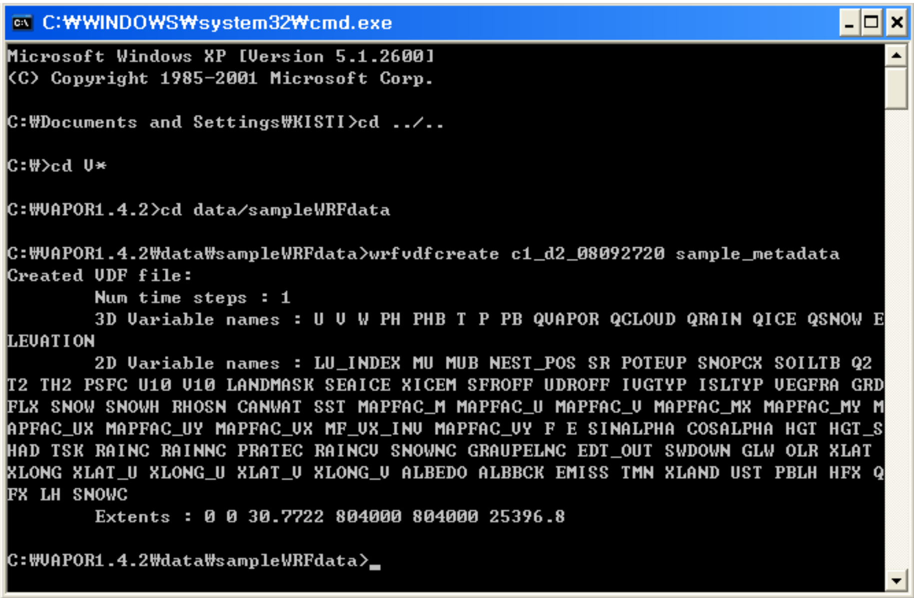
% *wrfvdfcreate wrf_output_data_files metadata_file.vdf*

where,

wrf_output_data_files is a list of one or more WRF output data files that you want to use.

metadata_file.vdf is the name that you will use for your metadata file.

The following is an example of creating a sample metadata file "sample_metadata.vdf" with a sample WRF data "c1_d2_08092720"



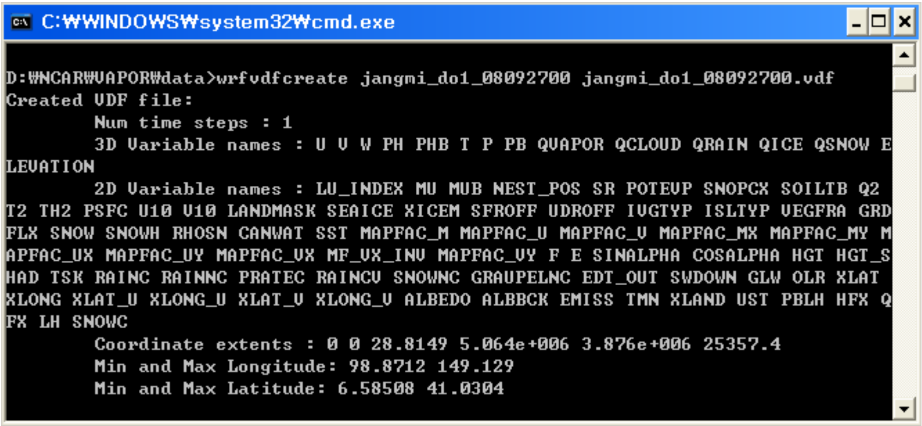
<Figure 3.2> Creating a metadata file

Note that a .vdf file is an XML file containing metadata describing an entire dataset (the output of a single simulation); for example, domain size, total number of time steps, and variable names.

Note that the default (specifying no options) works well in most cases. By default, all floating point 2D and 3D variables that use the spatial dimensions WEST_EAST, SOUTH_NORTH, BOTTOM_TOP, and their staggered versions, will be included in the VDC. Users may override the default, for example, specifying that a subset or superset of the variables or time steps will be included.

· Third, issue the command *wrf2vdf*.

% *wrf2vdf metadata_file.vdf wrf_output_data_files*



<Figure 3.3> Performing a data conversion

By default, the *wrf2vdf* command will convert all 3D and 2D variables that are specified in the VDC and will also create a new 3D variable, named "ELEVATION", that is needed during VAPOR visualization to interpolate data from the WRF grid to a Cartesian grid used for visualization and flow integration.

Note that *wrf2vdf* does most of the work, and may take a few minutes to convert a large WRF model data.

After issuing the above commands, *wrfvdfcreate* and *wrf2vdf*, all of the spatial variables in the specified WRF model output data files will be converted, for all the time steps in the files.

If you desire more control over the conversion process, there are many additional options that you can provide to *wrfvdfcreate* and *wrf2vdf*.

% *wrfvdfcreate [options] wrf_output_data_files metadata_file.vdf*

% *wrf2vdf [options] metadata_file.vdf wrf_output_data_files*

The options are described (with examples) in the *wrtvdfcreate* and *wrf2vdf* man pages. The man pages can be viewed on the VAPOR documentation website (http://www.vapor.ucar.edu/docs/reference-manuals/reference-manuals#CLIMANPAGES) and are also available using the "man" command on LINUX systems where VAPOR is installed.

You can see the description of *wrtvdfcreate* options in the command using the following command:

% *wrfvdfcreate* -h

```
C:₩WINDOWS₩system32₩cmd.exe

D:₩NCAR₩VAPOR₩data>wrfvdfcreate -h
Usage: wrfvdfcreate [options] wrf_ncdf_file... vdf_file
Usage: wrfvdfcreate [options] -startt time vdf_file
    OPTION           NUM_ARGS     DEFAULT
    ------           --------     -------
    -dimension         1          512x512x512
        Volume dimensions (unstaggered) expressed in grid points (NXxNYxNZ) if
        no sample WRF file is given
    -extents           1          0:0:0:0:0:0
        Colon delimited 6-element vector specifying domain extents in user
        coordinates (X0:Y0:Z0:X1:Y1:Z1) if different from that in sample WRF
    -startt            1          ""
        Starting time stamp, one of (time|SIMULATION_START_DATE|START_DATE),
        where time has the form : yyyy-mm-dd_hh:mm:ss
    -numts             1          0
        Maximum number of VDC time steps
    -deltat            1          0
        Seconds per VDC time step
    -varnames          1          ""
        Deprecated. Use -vars3d instead
    -vars3d            1          ""
        Colon delimited list of 3D variables to be extracted from WRF data
    -vars2d            1          ""
        Colon delimited list of 2D variables to be extracted from WRF data
    -dervars           1          ""
        Colon delimited list of desired derived variables. Choices are:
        PHNorm_: normalized geopotential (PH+PHB)/PHB, UVW_: 3D wind speed
        (U^2+V^2+W^2)^1/2, UV_: 2D wind speed (U^2+V^2)^1/2, omZ_: estimate of
        vertical vorticity, PFull_: full pressure P+PB, PNorm_: normalized
        pressure (P+PB)/PB, Theta_: potential temperature T+300, TK_: temp. in
        Kelvin (T+300)((P+PB))/100000)^0.286
    -level             1          2
        Maximum refinement level. 0 => no refinement (default is 2)
    -atypvars          1          U:V:W:PH:PHB:P:PB:T
        Colon delimited list of atypical names for U:V:W:PH:PHB:P:PB:T that
        appear in WRF file
    -comment           1          ""
        Top-level comment (optional)
    -mapprojection     1          ""
        A whitespace delineated list of PROJ4 +paramname=paramvalue pairs. If
        this is invalid, then time-varying extents will not be provided in the
        metadata
    -bs                1          32x32x32
        Internal storage blocking factor expressed in grid points (NXxNYxNZ)
        (default is 32)
    -nfilter           1          1
        Number of wavelet filter coefficients (default is 1)
    -nlifting          1          1
        Number of wavelet lifting coefficients (default is 1)
    -help              0          false
        Print this message and exit
    -quiet             0          false
        Operate quietly

Copyright 2007 The National Center for Atmospheric Research

Version: 1.4.3 ($Date: 2009/02/20 23:02:41 $) www.vapor.ucar.edu
```

<Figure 3.4> Options of the command wrfvdfcreate

Several additional derived variables may be calculated during the data conversion using *-dervars* option of the *wrfvdfcreate* command. The derived variables include:

PHNorm_: The normalized geopotential, (PH+PHB)/PHB

UVW_: The three-dimensional wind speed

UV_: The horizontal wind speed

omZ_: Approximate vertical vorticity

PFull_: The full pressure (P+PB)

PNorm_: Normalized pressure (P+PB)/PB

Theta_: The potential temperature T+300

TK_: Temperature in Kelvin=Theta*((P+PB)/100000)$^{0.286}$

You can also see the description of *wrf2vdf* options in the command window using the following command:

% *wrf2vdf* -h

```
C:₩WINDOWS₩system32₩cmd.exe

D:₩NCAR₩VAPOR₩data>wrf2vdf -h
Usage: wrf2vdf [options] metafile wrffiles...

    OPTION      NUM_ARGS     DEFAULT
    ------      --------     -------
    -varnames   1            ""
        Colon delimited list of variable names in WRF file to convert. The
        default is to convert variables in the set intersection of those
        variables found in the VDF file and those in the WRF file.
    -tsincr     1            1
        Increment between Vapor times steps to convert (e.g., 3=every third),
        from Vapor time step 0 (NOT CURRENTLY SUPPORTED!!!)
    -numts      1            -1
        Maximum number of time steps that may be converted. A -1 implies the
        conversion of all time steps found
    -tsstart    1            ""
        Starting time stamp for conversion (default is found in VDF)
    -tsend      1            ""
        Last time stamp to convert (default is latest time stamp)
    -level      1            -1
        Refinement levels saved. 0=>coarsest, 1=>next refinement, etc.
        -1=>finest
    -noelev     0            false
        Do not generate the ELEVATION variable required by vaporgui.
    -help       0            false
        Print this message and exit.
    -debug      0            false
        Enable debugging.
    -quiet      0            false
        Operate quietly (outputs only vertical extents that are lower than
        those in the VDF).

Copyright 2007 The National Center for Atmospheric Research

Version: 1.4.3 ($Date: 2009/02/20 23:02:41 $) www.vapor.ucar.edu
```

<Figure 3.5> Options of the command wrf2vdf

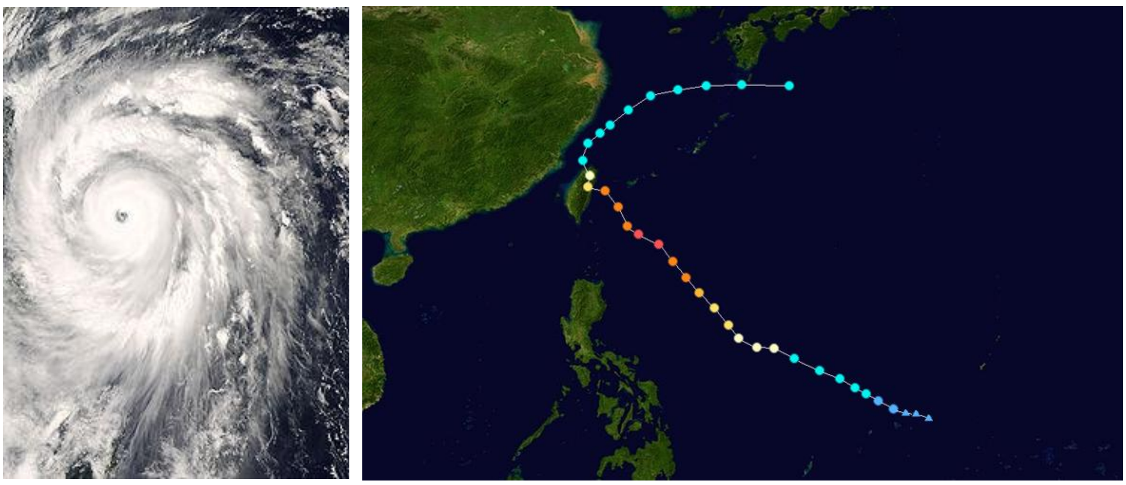
3.2 Sample WRF model output data for typhoon

3.2.1 Case: Typhoon Jangmi

Typhoon Jangmi is recognized as the 15th Severe Tropical Storm as well as the 10th typhoon of the 2008 Pacific typhoon season by the JMA (Japan Meteorological Agency).

· Formed	September 23, 2008
· Dissipated	October 1, 2008
· Highest wind	215 km/h (130 mph) (10-minute sustained) 260 km/h (160 mph) (1-minute sustained) 325 km/h (200 mph) (gusts)
· Lowest pressure	905 hPa
· Fatalities	1 direct, 1 indirect, 2 missing
· Damage	$240.4 million (2008 USD)
· Areas affected	Philippines, Taiwan, China

Taiwan was affected by Typhoon Jangmi. Taiping Mountain in Yilan County registered 692 mm of precipitation according to the 0415 UTC advisory while Nangang in Taipei City in northern Taiwan reported 463mm of precipitation. The highest recorded rainfall was registered in Tatung Town at 1,124mm.



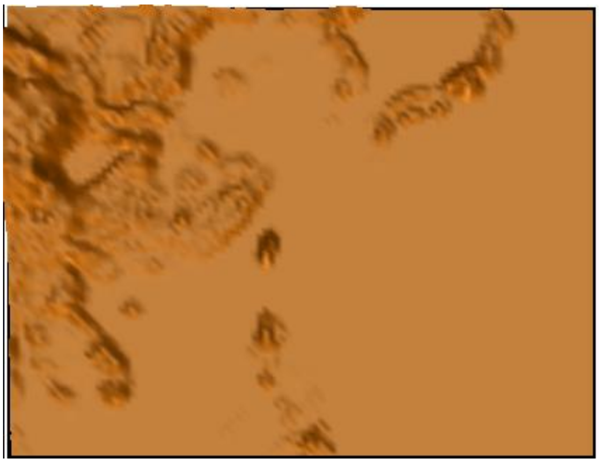
<Figure 3.6> Typhoon Jangmi at Category 5 intensity and storm path

3.2.2 Preparing a WRF model output data of Typhoon Jangmi

The WRF model setup for simulating Typhoon Jangmi is as follows:

Domain		1 (fixed domain)	2 (moving domain)
Horizontal Resolution		12 km	4 km
No. of Grid points		424 X 325	202 X 202
Vertical Layer		35	35
Time	start	2008-09-26-12:00:00	
	end	2008-10-01-12:00:00	
	interval	3 hr	1 hr
	steps	41	121

The terrain surface of each domain represented by VAPOR is like the following:



(a) Domain 1 (at all time steps)

(b) Domain 2 (at time step=41)

<Figure 3.7> Terrain surfaces in model domains (exaggerated vertically)

In this User's guide, (fixed) Domain 1 WRF model simulation data of Typhoon Jangmi is used to get the longitude/latitude values of lower-left and upper-right of the full domain for making a georeferenced terrain image. To produce the example figures showing the VAPOR capabilities, (moving) Domain 2 data is used.

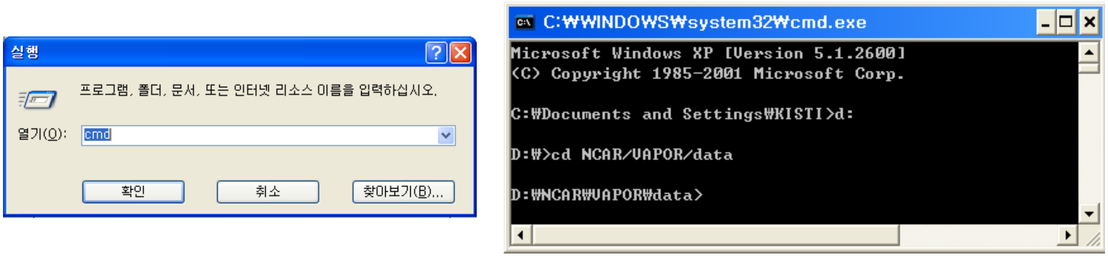
3.3 Preparing a georeferenced terrain image

VAPOR provides a command-line utility (*tiff2geotiff*) to make a georeferenced terrain image file (geotiff) from a general satellite image file (tiff).

The simplest way to create a georeferenced terrain image file is as follows:

· First, open the command window and change the current directory to the directory that includes the WRF output data.

Click on "Windows Start" --> "Run" --> "cmd"



(a) execution dialog (b) command window

<Figure 3.8> Open the command window

· Second, read the Min./Max. Longitude/Latitude values from the standard output displayed on the screen when you issue the command *wrfvdfcreate*. For details, see Section 3.1 and <Figure 3.3>.

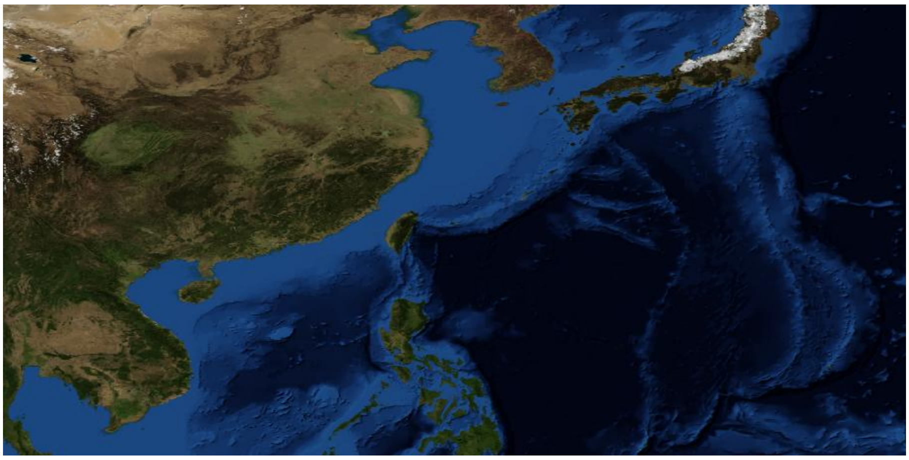
· Third, download a satellite image from the NASA Web mapping service using the Min./Max. Longitude/Latitude values (as *llx, lly, urx, ury*) in the following URL that you submit in your Web browser (note that all three lines of text are submitted in one line as one URL to the browser):

> http://www.nasa.network.com/wms?request=GetMap&version=1.3&
> layers=|bmng200401&styles=default&bbox=*llx,lly,urx,ury*&width=1024&
> height=512&format=image/tiff

where,

llx, lly are lower-left longitude and latitude of a domain
urx, ury are upper-right longitude and latitude of a domain

From the <Figure 3.3>, we can set the values of *llx, lly, urx, ury* to 98.8712, 6.58508, 149.129, 41.0304, respectively.



<Figure 3.9> Satellite image downloaded from the NASA Website
(Longitude: 98.8712,-149.129, Latitude: 6.58508,-41.0304)

· Finally, issue the command *tiff2geotiff* to convert the TIFF image file into a geotiff image file.

% *tiff2geotiff -4 "proj4_str" -n "llx lly urx ury" input_image_file output_image_file*

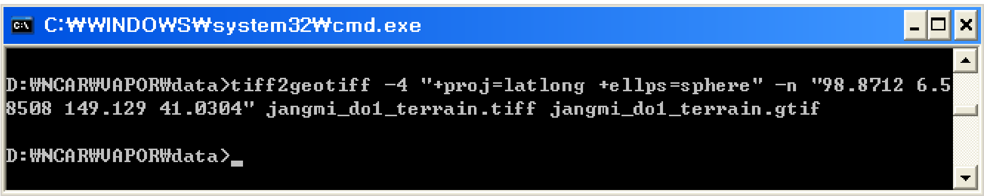
where,
-4 "proj4_str" specifies the map projection to use. You must substitute a valid PROJ4 projection string for "proj4_str". In most cases "proj4_str" will be
"+proj=latlong +ellps=sphere". (see below for other values for this string)
-n "llx lly urx ury" identifies the longitude/latitude extents to install;
four longitude and latitude values must be provided in the order:
lower-left longitude, lower-left latitude
upper-right longitude, upper-right latitude
input_image_file is the satellite image file that you downloaded from NASA Website (format: tiff)
output_image_file is the name that you will use for your terrain image file (format: geotiff)

For satellite images, the data is on a latitude/longitude grid, so set *"proj4_str"* to the string "+proj=latlong +ellps=sphere".

If your data uses a different type of map projection, you should set the value of *"proj4_str" to* the appropriate string for the projection you are using. For more information about the meaning of the PROJ4 string, "*proj4_str*", visit the PROJ.4-Cartographic Projections Library Website:

http://trac.osgeo.org/proj

To convert the tiff image from the NASA web mapping service into a georeferenced terrain image for the example case (Typhoon Jangmi), referring to <Figure 3.3>, we can set the values of "*llx lly urx ury"* to "98.8712 6.58508 149.129 41.0304".



<Figure 3.10> Using tiff2geotiff to convert a tiff file into a geotiff file

VAPOR provides another command-line shell script *getWMSImage.sh* that will make a georeferenced terrain image by retrieving the image from the NASA website and inserting the appropriate geo-referencing information into the file.

The *getWMSImage.sh* is a bash-shell script, so it operates in Unix environments. It operates in two modes: default mode and expert. The default mode is intended for the user who merely wants to acquire NASA Blue Marble or Landsat imagery. It uses the NASA WorldWind (worldwind.arc.nasa.gov/java) site to acquire the imagery. The expert mode is intended for using the script to fetch an image from any OGC WMS server. You should not use the expert mode unless you are familiar with Web mapping services.

For details, see section 2 of the VAPOR/WRF Data and Image Preparation Guide.

3.4 Preparing georeferenced 2D plots using NCL

VAPOR provides a capability to load 2D plots created by NCL into the 3D scene. However, the output file format must be a tiff or geotiff to be used in VAPOR.

There are various NCL scripts, provided by Cindy Bruyere, that can be used to plot WRF-ARW data in NCL. These scripts are available at
http://www.mmm.ucar.edu/wrf/OnLineTutorial/Graphics/NCL/NCL_examples.htm
Users can use these NCL scripts, making five minor modifications, in order to make a georeferenced 2D plot.

Modify your NCL script as follows:

First, load supporting libraries.
load " ...path_to_supporting_scripts.../wrf2geotiff.ncl "

Second, initialize the geotiff-capture process.
wrf2gtiff=wrf2geotiff_open(wks)

Third, set times.
times=wrf_user_list_times(wrfFile)

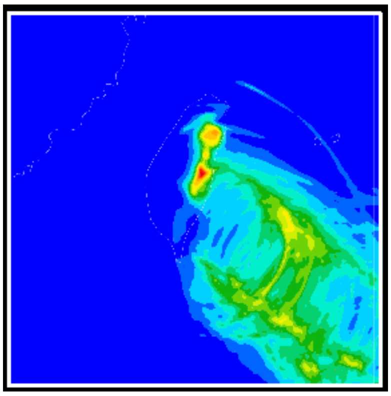
Fourth, write the plot to the geotiff file.
wrf2geotiff_write(wrf2gtiff, wrfFile, times(it), wks, plot, True)

If the last element of *wrf2geotiff_write* is "True", you will get a cropped image. If is "False", you will get an uncropped image.

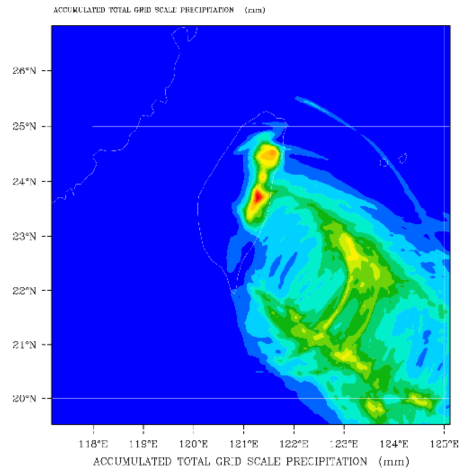
Finally, close the geotiff file.
wrf2geotiff_close(wrf2gtiff, wks)

To get a georeferenced 2D image for our example case (Typhoon Jangmi), you can make an accumulated precipitation by making these modifications to an example NCL script (e.g. "*wrf_Acc_Precip.ncl* ").

The 2D images created by "*wrf_Acc_Precip.ncl*" are as follows:



(a) cropped image



(b) uncropped image

<Figure 3.11> Georeferenced 2D images created by NCL

For details, see the Appendix; also consult section 3 of the [VAPOR/WRF Data and Image Preparation Guide].

Chapter 4: VAPOR Basics

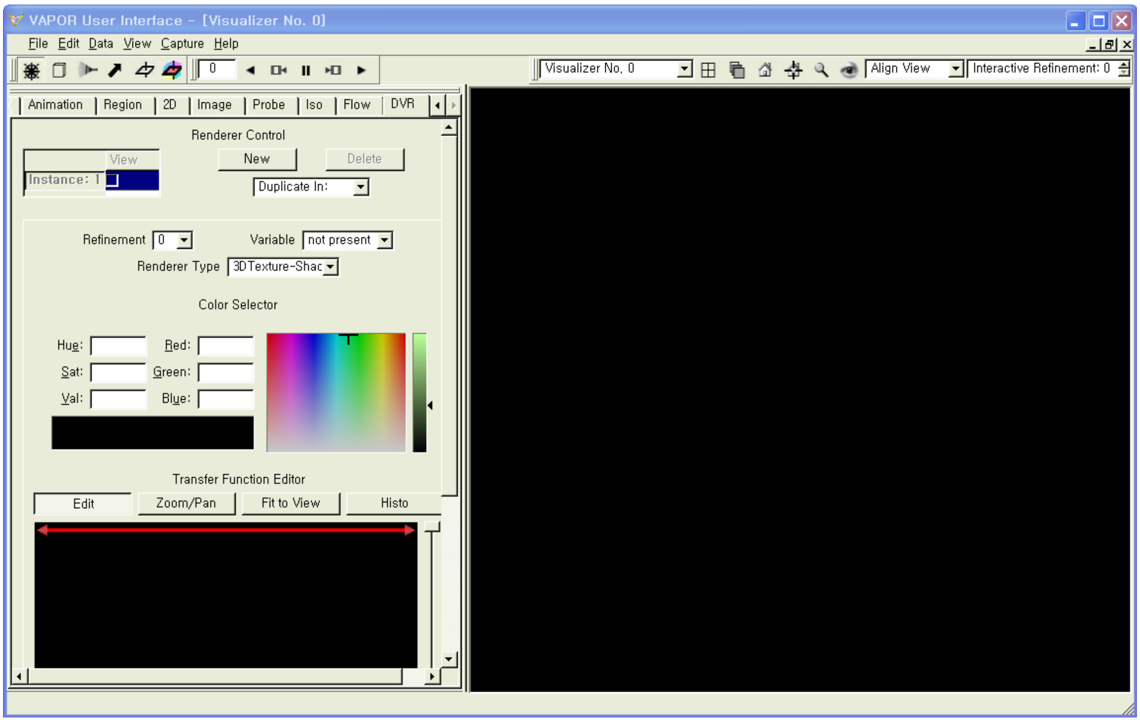
4.0 VAPOR Example data

To run VAPOR you will need to have access to data that has been converted to a VAPOR VDC. You can follow the instructions in Chapter 3 to convert an existing WRF-ARW dataset. In addition there are two example converted WRF datasets available on the VAPOR website at http://www.vapor.ucar.edu/page/vapor-download#Example. A low-resolution version of typhoon Jangmi is available, named "jangmi_lowres" and a low resolution of a cold spell in April 2007 in the state of Georgia, US, is named "april_lowres". We shall refer to both of these examples in the following.

4.1 Launching VAPOR

On Windows system where VAPOR has been installed, you can start VAPOR just double-clicking on the "vaporgui" icon on the desktop, or type "vaporgui" from a console window.

The initial VAPOR GUI Window is like the following:

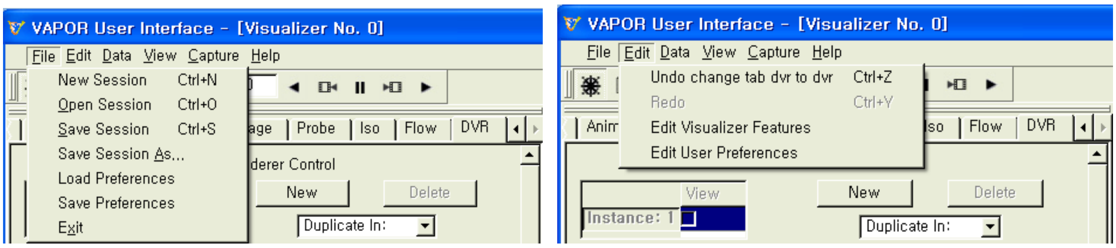


<Figure 4.1> Initial VAPOR GUI Window

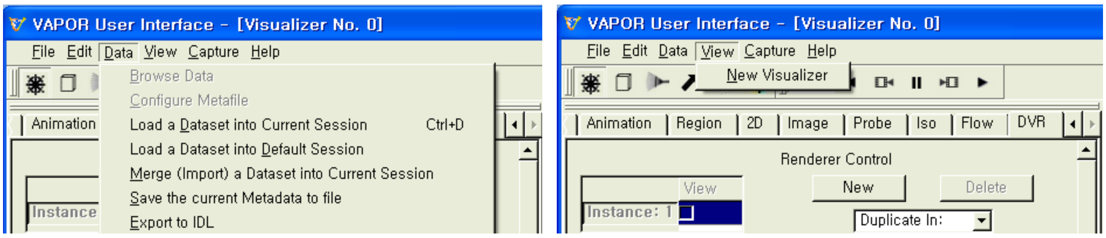
The right side of the VAPOR GUI Window is the "visualizer window", where the 2D and 3D graphics images are displayed. The left side of the VAPOR GUI Window has a set of tabs across the top, labeled "**View**", "**Animation**", "**Region**", "**2D**", "**Image**", "**Probe**", "**Iso**", "**Flow**", and "**DVR**". The function of each tab will be explained (with examples) in Chapter 5.

On the left-top of the VAPOR GUI Window, there is main menu including "**File**", "**Edit**", "**Data**", "**View**", "**Capture**", "**Help**". Each menu has submenu like the following:

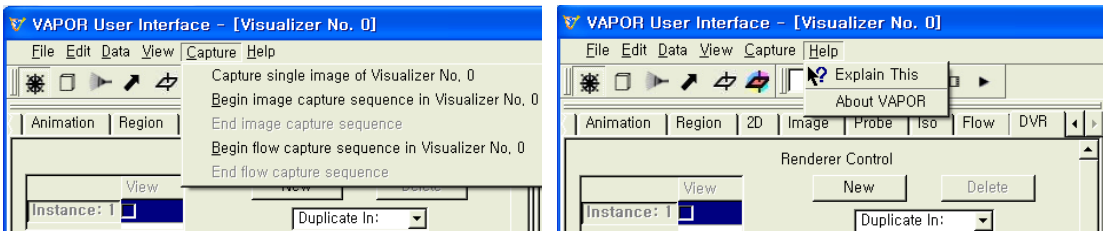
"File" and "Edit" menus



"Data" and "View" menus



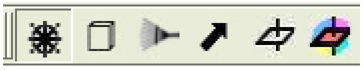
"Capture" and "Help" menus



<Figure 4.2> Seven main menus of the VAPOR GUI Window

To assist users in modifying visual features in the 3D scene, VAPOR has six mouse mode buttons above the tabs at the top-left of the VAPOR GUI Window.

Under the main menu, there are six icons representing the six mouse mode buttons like the following:



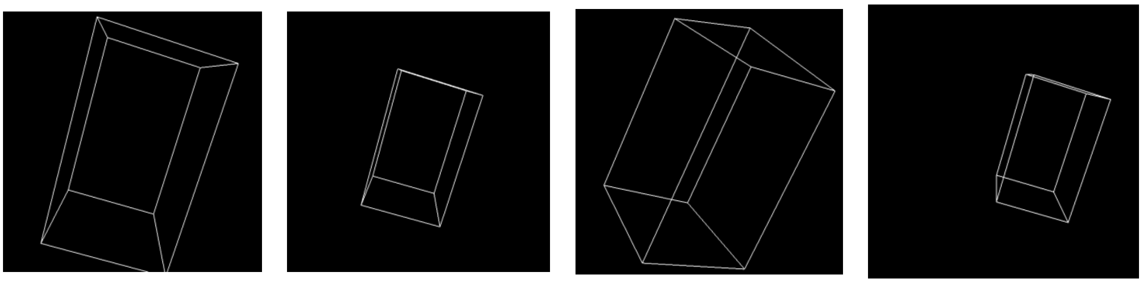
<Figure 4.3> VAPOR six mouse mode buttons

From the left, the icons represent "**Navigation Mode**", "**Region Select Mode**", "**Rake Mode**", "**Data Probe and Contour Mode**", "**2D Mode**", and "**Image Mode**", respectively. Clicking the buttons enables users to manipulate a shape being seen in the visualizer. These buttons are associated with the following manipulation modes:

1. "**Navigation Mode**" is used to rotate, zoom and translate the viewer position in the scene. Spin animation is also available in navigation mode.
2. "**Region Select Mode**" is used to manipulate the box-shaped region which is used for volume rendering, isosurfaces, and flow integration.
3. "**Rake Mode**" is used to specify a "rake" box, controlling the position of flow seed points.
4. "**Data Probe and Contour Mode**" is used to control the location and size of the probe/contour plane tool.
5. "**2D Mode**" is used to position a plane of two-dimensional data in the scene.
6. "**Image Mode**" used to manipulate image location and size; i.e. it is used for positioning terrain images and data plot images.

In the "**Navigation Mode**", users can rotate, move, and resize the graphic object in the visualizer manipulating the mouse buttons like the following:

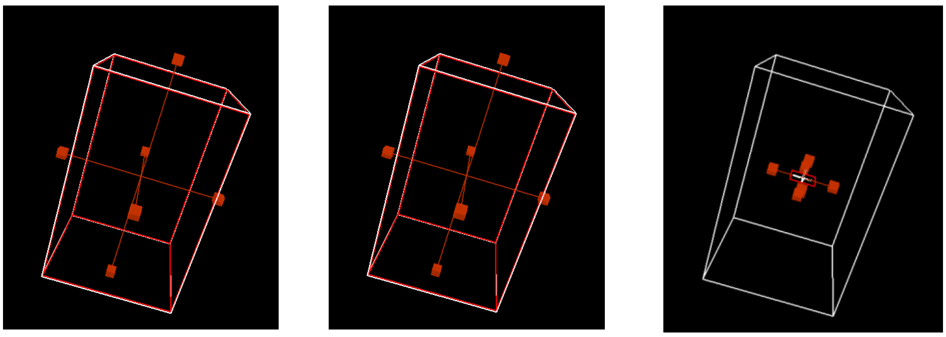
function	manipulation
zoom-in	drag the mouse up while pressing right mouse button
zoom-out	drag the mouse down while pressing right mouse button
rotate	drag the mouse left, right, up, down while pressing left mouse button
move	drag the mouse left, right, up, down while pressing middle mouse button



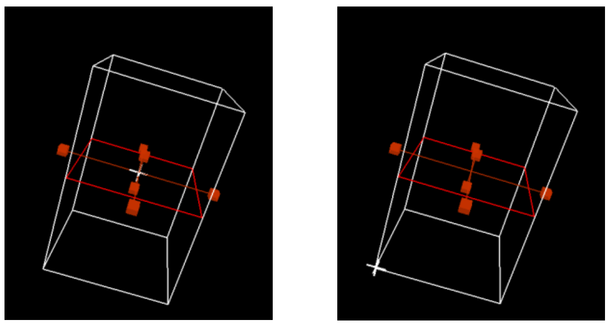
(a) zoom-in (b) zoom-out (c) rotate (d) move

<Figure 4.4> Navigation Mode

Modes other than "**Navigation Mode**" are obtained clicking the "**Region Select Mode**", "**Rake Mode**", "**Data Probe and Contour Mode**", "**2D Mode**", and "**Image Mode**". Using these buttons, users can see each manipulator for the selected mode like the following:



(a) Region Select (b) Rake (c) Data Probe

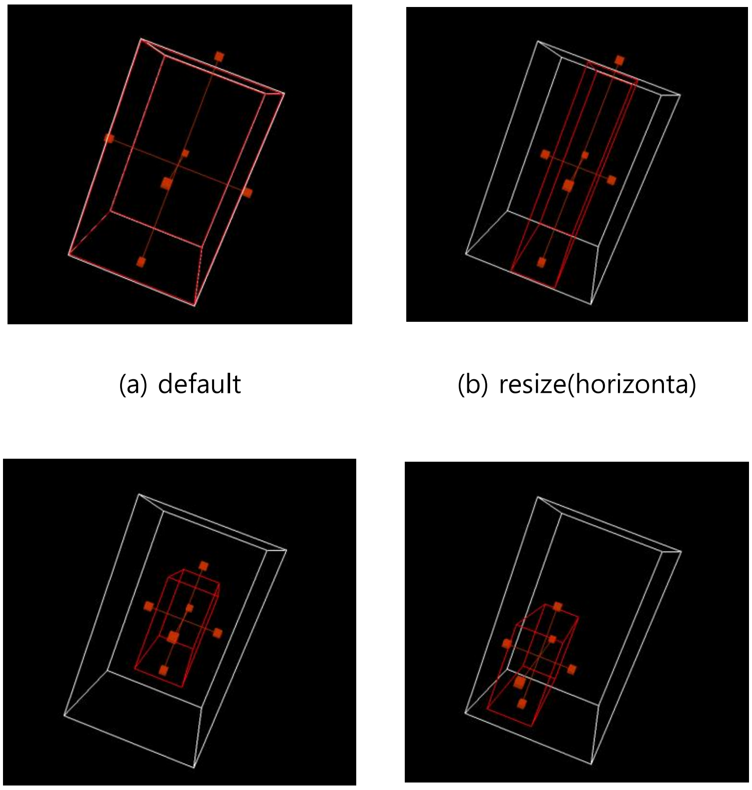


(d) 2D (e) Image

<Figure 4.5> Manipulators of the five mouse modes

If users click with the mouse button on the handle at the manipulator, the handle will change color from red to yellow. Users can move or resize the manipulator box in the visualizer, by pressing and dragging the mouse buttons like the following:

function	manipulation
move	press left mouse button on the handle of the manipulator, drag the handle (left, right, up, down), release the handle
resize	press right mouse button on the handle of the manipulator, drag the handle (left, right, up, down), release the handle



(a) default (b) resize(horizonta)

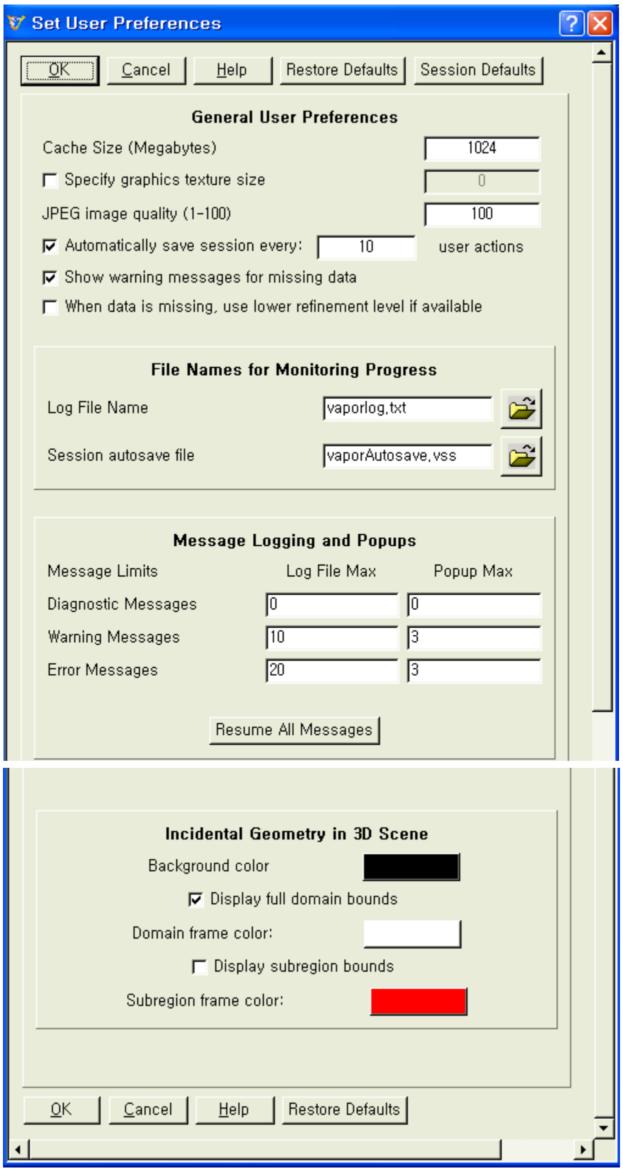
(c) resize(vertical) (d) move

<Figure 4.6> Manipulation of box extents

4.2 Setting User Preferences

Before loading a volume dataset, it is useful to set up your user preferences, which are settings that apply just to your individual system and usage requirements. Most of the defaults will work fine; however, one important preference that users may want to change is the data cache size. You will get the best performance from VAPOR if the cache size is about equal to the physical memory available to you on the computer you are using.

To set up your own user preferences, click "Edit" --> "Edit User Preferences". Set the "Cache Size (Megabytes)" to a value appropriate to the computer you are using, to get the best performance from VAPOR.

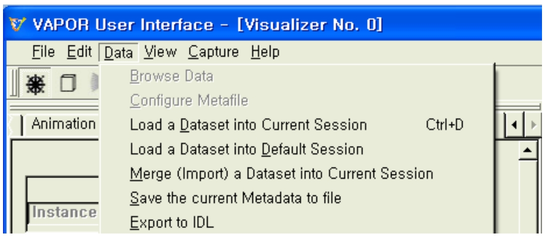


<Figure 4.7> User Preferences Panel

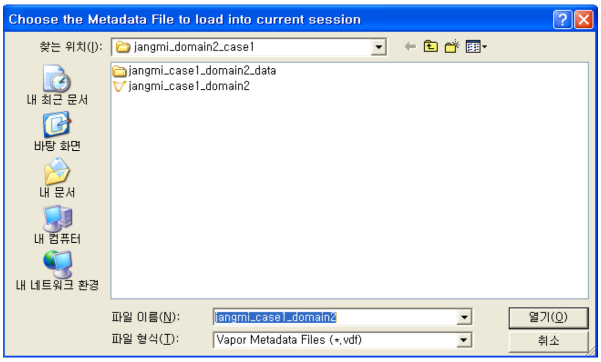
Users can change the background color (default, black), the domain frame color (default, white) and the subregion frame color (default, red) from default color to the color desires. Many other defaults can be set using this panel.

4.3 Specifying a Volume Dataset

Click "Data" at the top of VAPOR GUI Window. You will see a list of various commands that can be applied to your volume dataset. Click "Load a Dataset into Current Session". You will see a file selection dialog, asking you to "Choose the Metadata File to load into current session" like the following:



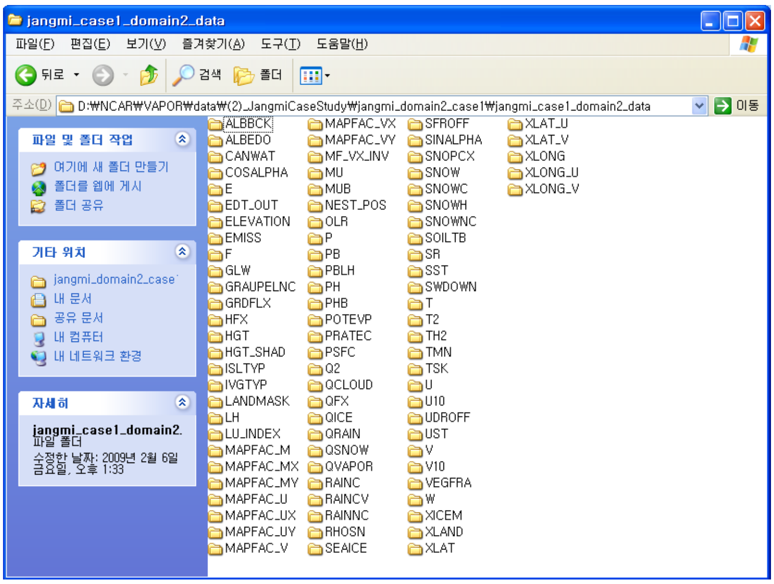
(a) Data submenu



(b) Metadata file selection dialog

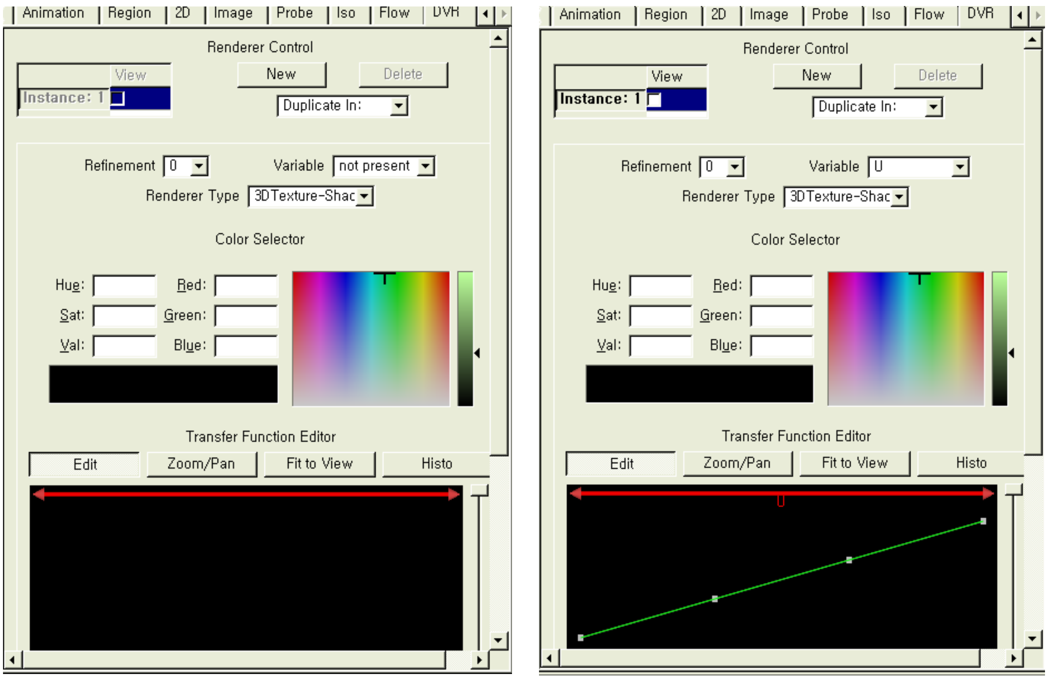
<Figure 4.8> Loading a metadata file

Metadata file selection dialog enables you to read a file of type ".vdf", which stands for "Vapor Data Format". These are Metadata files, not data files: .vdf files contain descriptions of data. Typically the referenced data is in many different files like the following:



<Figure 4.9> Example of referenced data

After loading a metadata file (e.g. "**jangmi_lowres.vdf**"), you can see changes in the DVR control panel like the following:

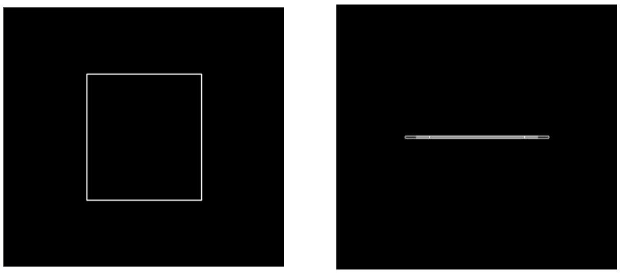


(a) Before loading a metadata file (b) After loading a metadata file

<Figure 4.10> Differences in DVR control panel

If you load the example metadata file "april_lowres.vdf", you can see a white rectangle in the visualizer window as in Figure 4.11(a). The left panel is actually the top view of a slab shaped region enclosing the data. This data volume is very flat, about 1000km in each of the two horizontal dimensions, and about 1.6km high.

You can also see the view of region from side as in Figure 4.11(b). Click and drag your left mouse in the visualizer window and you can rotate the rectangle you are viewing.



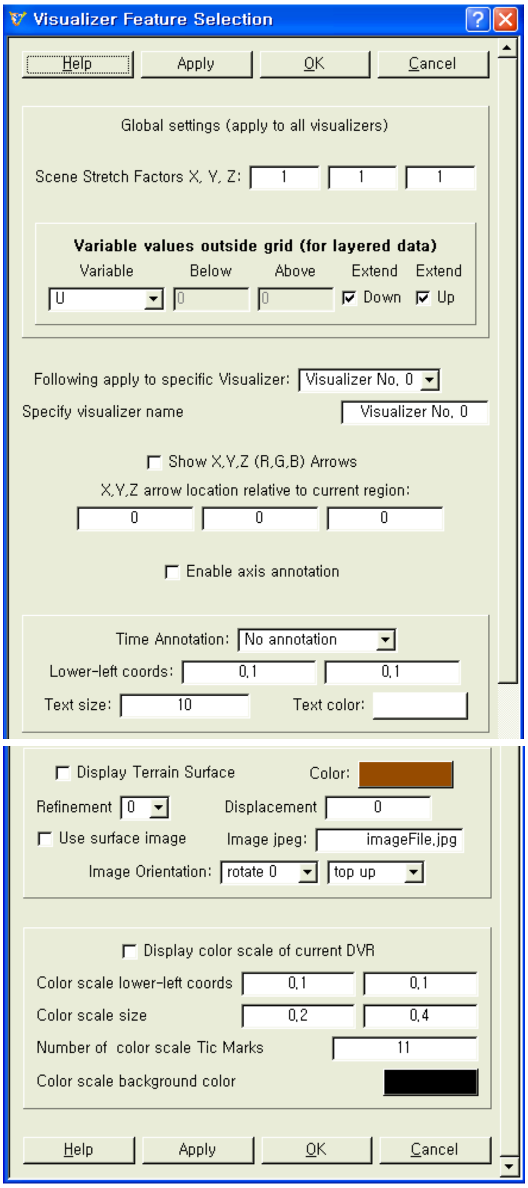
(a) view from top (b) view from side

<Figure 4.11> View of region

4.4 Setting Visualizer Features

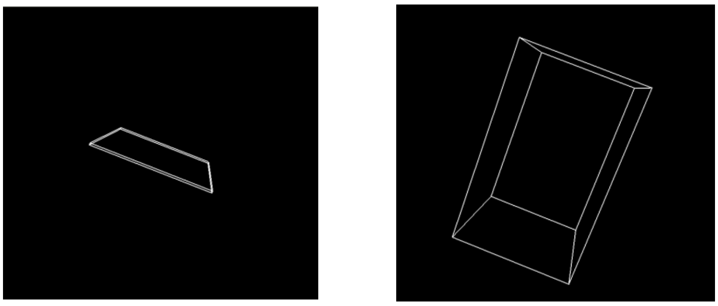
In general, WRF data volumes are very flat as you can see in Figure 4.11(b). This WRF model has about several thousand km in each of the two horizontal dimensions, and about several km high. In order to effectively visualize this volume data in 3D, you need to stretch this volume in the vertical direction, exaggerating the height.

To set up visualizer features, click on "Edit" --> "Edit Visualizer Features".



<Figure 4.12> Visualizer Feature Selection Panel

Set the "Scene Stretch Factors X, Y, Z:" to 1, 1, 50. Click "Apply" and you will see that the data volume has stretched vertically.



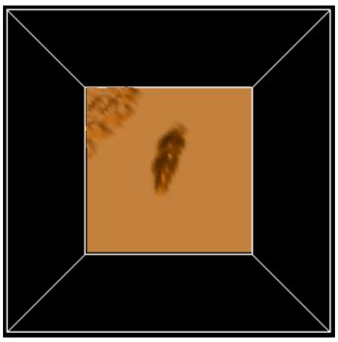
(a) Before stretching (b) After stretching

<Figure 4.13> Vertical scene stretching effect

There are a number of other capabilities controlled by this panel. You can display the terrain as a surface and apply a terrain image onto that surface. For correctly visualizing terrain images in the scene we recommend using the capabilities of the Image panel, instead of applying a terrain image as a visualizer feature. Section 5.2 shows how to load terrain images using the "Image" panel.

The terrain images specified in the visualizer features panel require that the image you use has already been prepared to coincide with the extents of the WRF data. To set up a terrain image in the visualizer features, check the box labeled "Display Terrain Surface". Click "OK" at the top of bottom of the panel. You will see the region box in the visualizer as in Figure 4.14.

Note: you may need to set the value of the "Displacement" in the visualizer features panel to a positive value, such as 80, with the typhoon Jangmi data. The terrain surface will not display if it is too low to fit inside the current region.

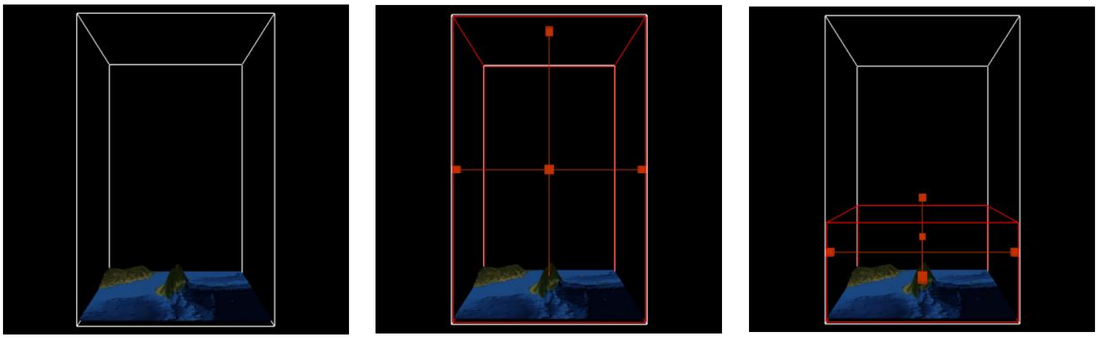


<Figure 4.14> Display of terrain surface

4.5 Setting Visualization Region

Figure 4.15(a) shows the full domain. WRF model users are mostly interested in the weather near the terrain surface, so you can restrict your attention to the region near the bottom of the volume.

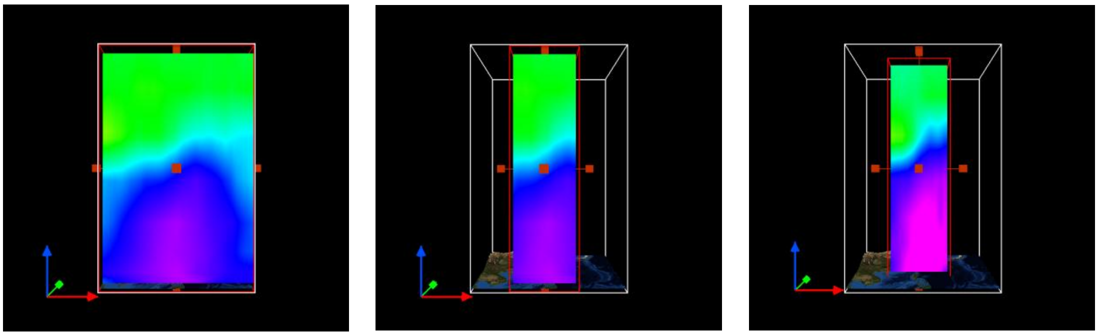
To set up this region in the visualizer, click the cube symbol near the top left of the VAPOR GUI Window, on the button identified in its tool-tip as "Region Select Mode". You will see that the volume in the visualizer window becomes enclosed by red lines, with cube-shaped handles on each side (Figure 4.15(b)). You have already seen this region select box in Figure 4.5(a). Grab the handle on the top of the region select box and drag it down so that the region is nearer the ground surface.



(a) default region (b) region select mode (c) modified region in Z

<Figure 4.15> Setting the visualization region vertical extents

You can also change the region of interest in other directions, by right-mouse dragging other handles of the region select box, as in the following:



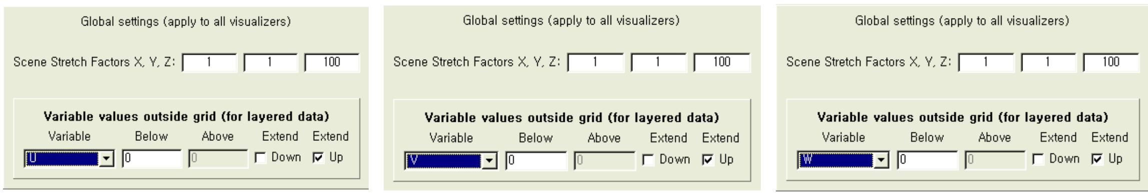
(a) default region (b) modified region in X (c) modified region in Y

<Figure 4.16> Setting visualization region in horizontal direction

4.6 Other Useful Settings

In steady flow visualization (Chapter 5), you may see a warning message indicating that the flow variables (U, V, W) are not zero at the bottom of the data. This can have the undesirable effect of causing wind to appear to flow through and below the terrain.

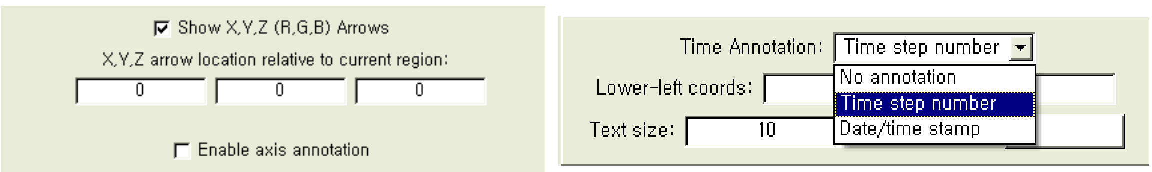
To fix this issue, click on "Edit"--> "Edit Visualizer Features". There is a frame labeled "Variable values outside grid (for layered data)" near the top of the Visualizer Features Panel (Figure 4.12). Select the variable "U", uncheck the check-box labeled "Extend Down", and specify the value of 0 in the box labeled "Below". Do the same for the variables "V" and "W". Click "OK". Now the wind field will be always equal to zero below the terrain.



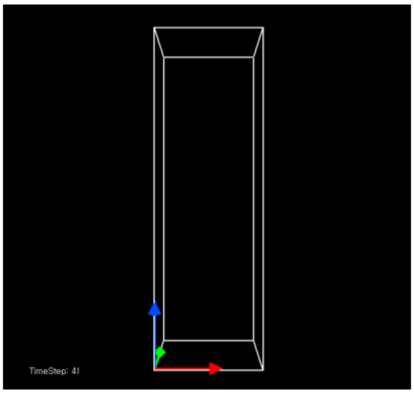
(a) variable U (b) variable V (c) variable W

<Figure 4.17> Specifying variable values outside grid

Sometimes, it will be helpful to see the coordinate axes and time annotation in the visualizer. To set this up, click on "Edit" --> "Edit Visualizer Features". Click the check-box labeled "Show X,Y,Z(R,G,B) Arrows" and set "Time step number" in the time annotation selection box. You can also select “Date/Time Stamp” to display the WRF time stamp in the scene.



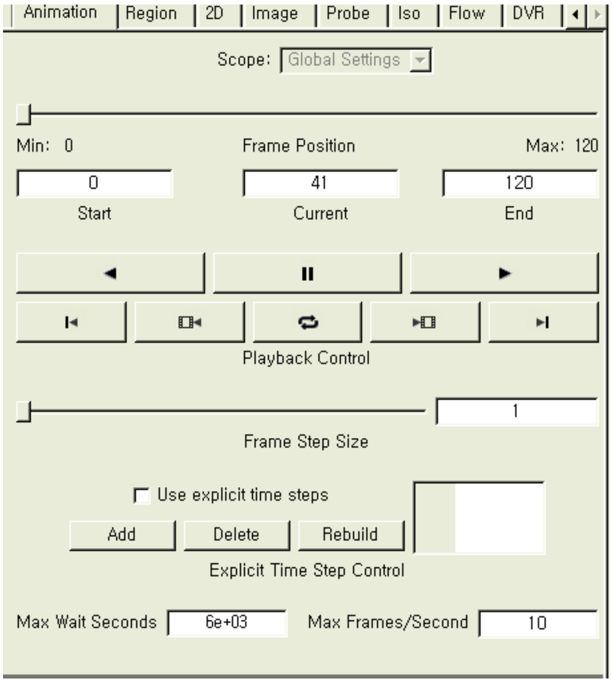
(a) specify X, Y, Z axes (b) specify time annotation



(c) results

<Figure 4.18> Specifying X, Y, Z axes and time annotation

To select the specific time step to be displayed in the visualizer window, click the "Animation" tab. Set "Current" in the "Frame Position" to the time step you want to display. By clicking the play button, you can display all the images. If you want to play the images faster (or more slowly), increase (or decrease) the value of "Max Frames/Second". (Default value = 10)



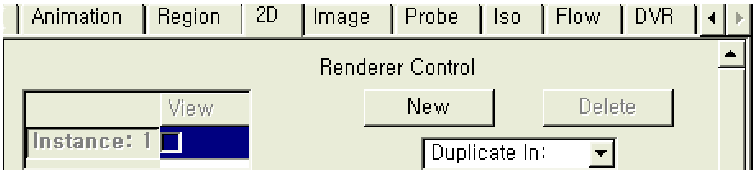
<Figure 4.19> Animation panel

Chapter 5: Visualization

5.1 Enabling rendering in Visualizers

5.1.1 Single Visualizer

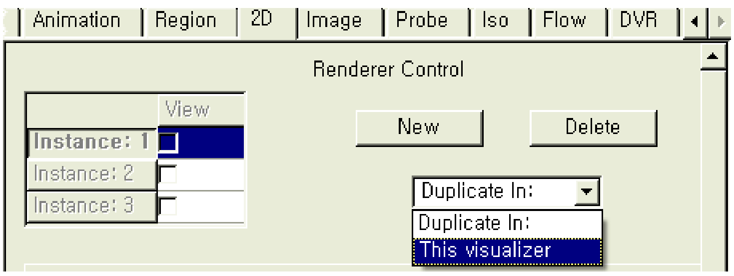
"**2D**", "**Image**", "**Probe**", "**Iso**", "**Flow**", and "**DVR**" tabs control the rendering of images in the visualizers. Each of these tabs has a "Render Control" section near the top of the panel, appearing as follows:



<Figure 5.1> Renderer control section

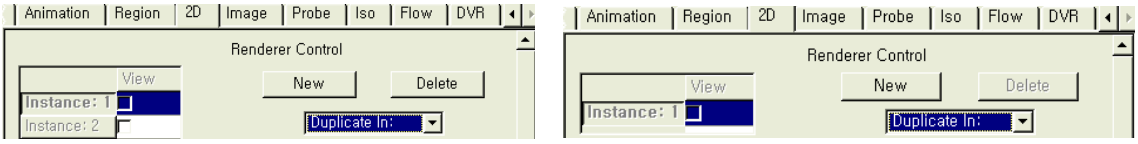
Using the "Renderer Control" section, you can control any number of renderers in the current visualizer window. Note that you can control the renderers only after you have loaded a dataset.

In the current visualizer window, additional renderers can be created by clicking the "New" button or "This visualizer" in the "Duplicate In:" selection box.



<Figure 5.2> Creating renderers

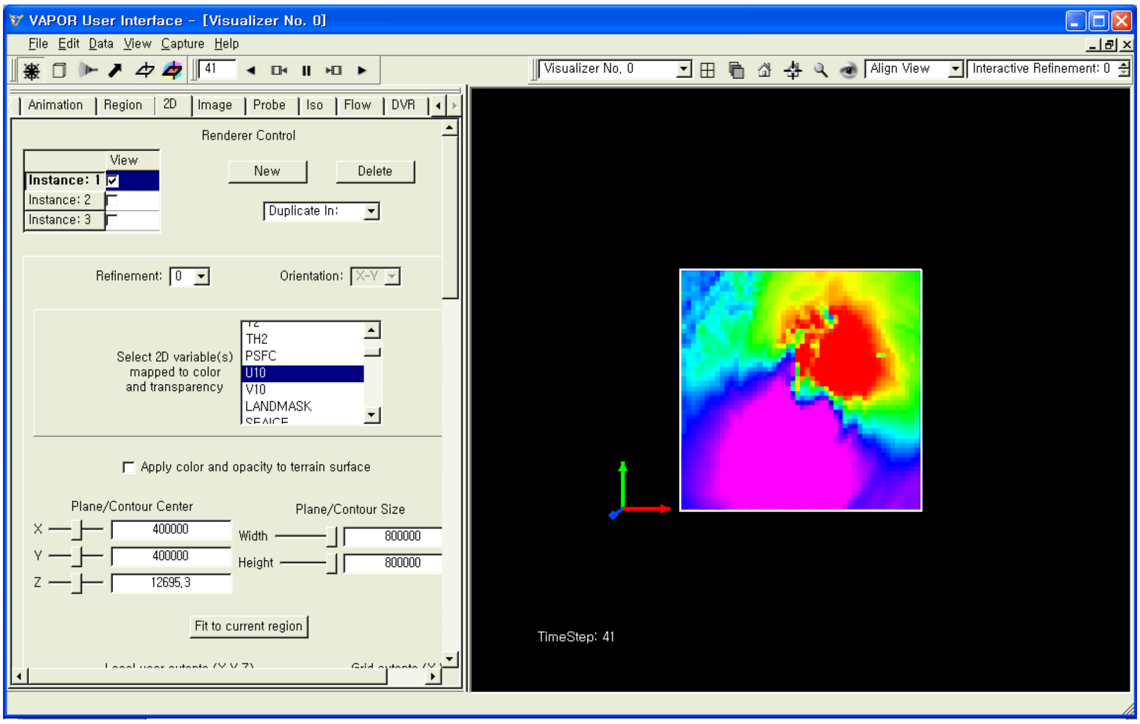
The active renderer instance number is indicated by boldface text on "Instance: #" button, and it appears depressed relative to adjacent instance buttons.
Note that only after creating a new renderer is the "Delete" button is activated. Unnecessary renderers can be deleted by clicking the "Delete" button.



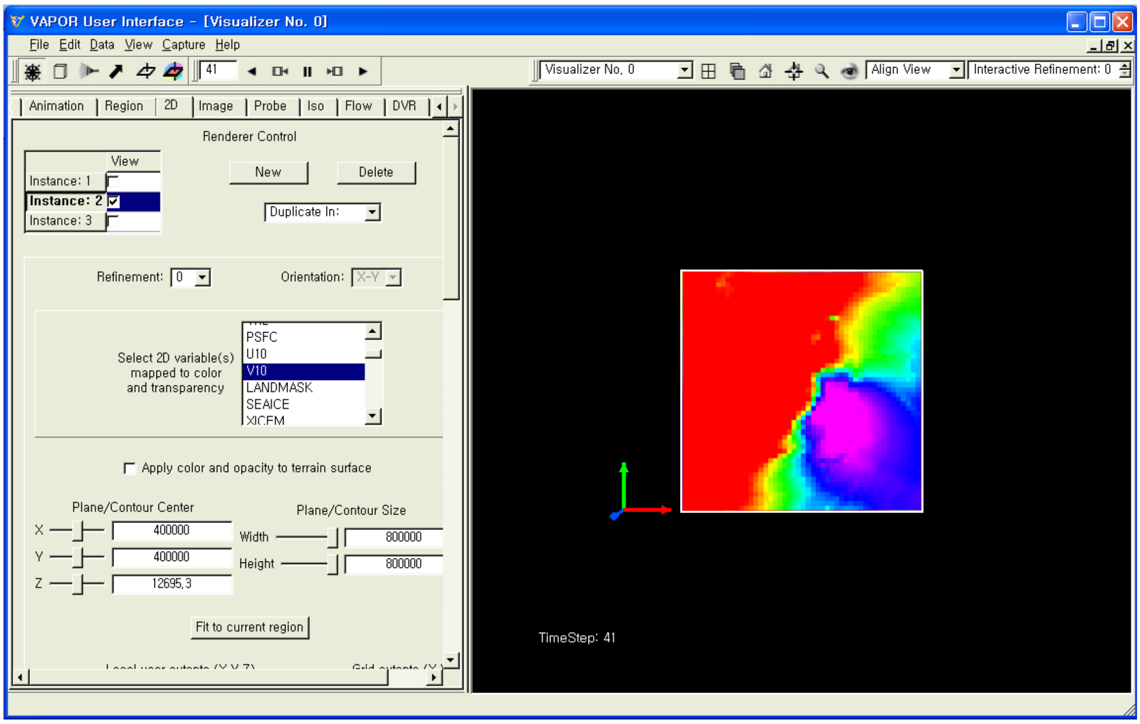
(a) Before clicking the "Delete" button (b) After clicking the "Delete" button

<Figure 5.3> Deleting renderers

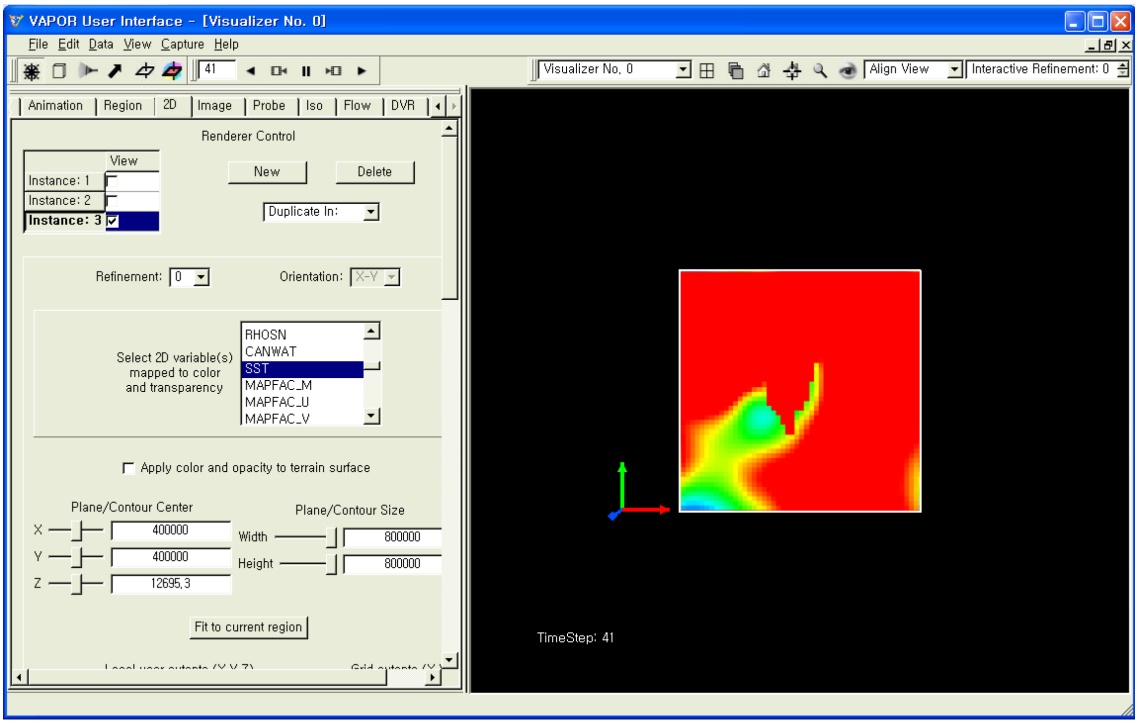
The different renderers are identified as "Instance: 1", "Instance: 2", "Instance: 3", etc. When you click in the check-box by the "Instance: #", the specified renderer (#) is enabled; i.e. the renderer will start drawing in the current visualizer window according to the parameters displayed in the current tab, below the "Renderer Control" section. As an example, click on the 2D tab, then create two new renderer instances. Select the variables U10, V10, and SST in the different instances. Change the Plane/Contour Z coordinate of these instances so that they do not coincide. You can display three different 2D variables - U10, V10, and SST in the current visualizer, individually or simultaneously.



(a) Render instance 1: Visualization of 2D variable U10 at time step=41

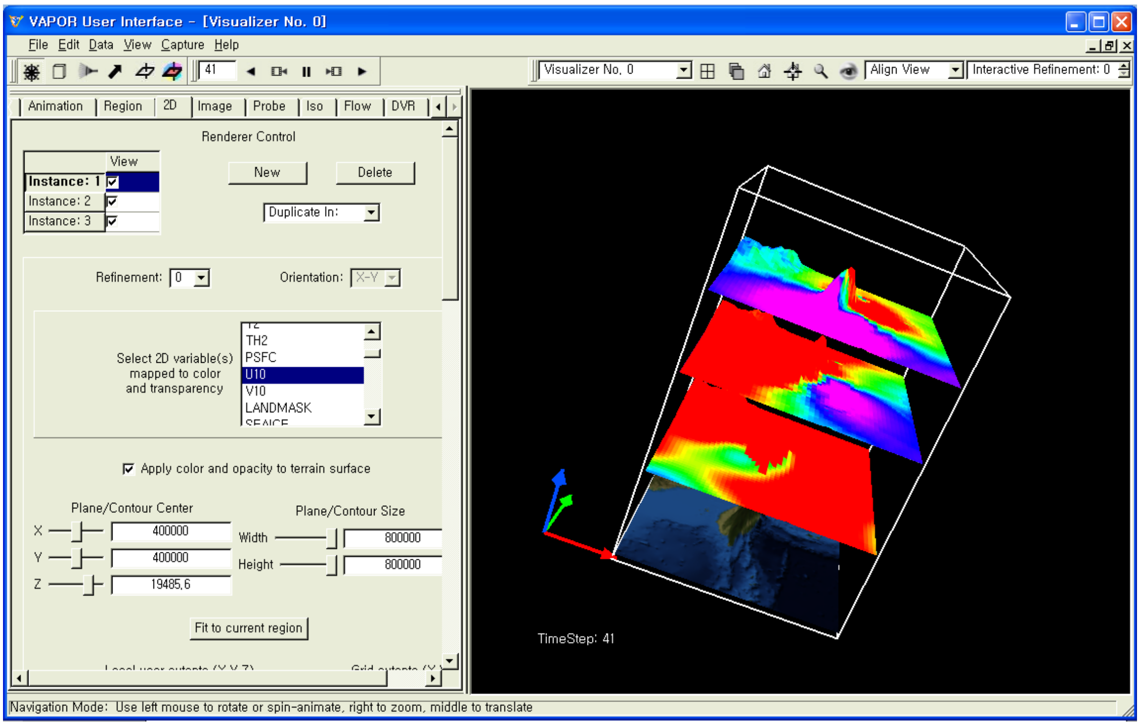


(b) Renderer instance 2: Visualization of 2D variable V10 at time step=41

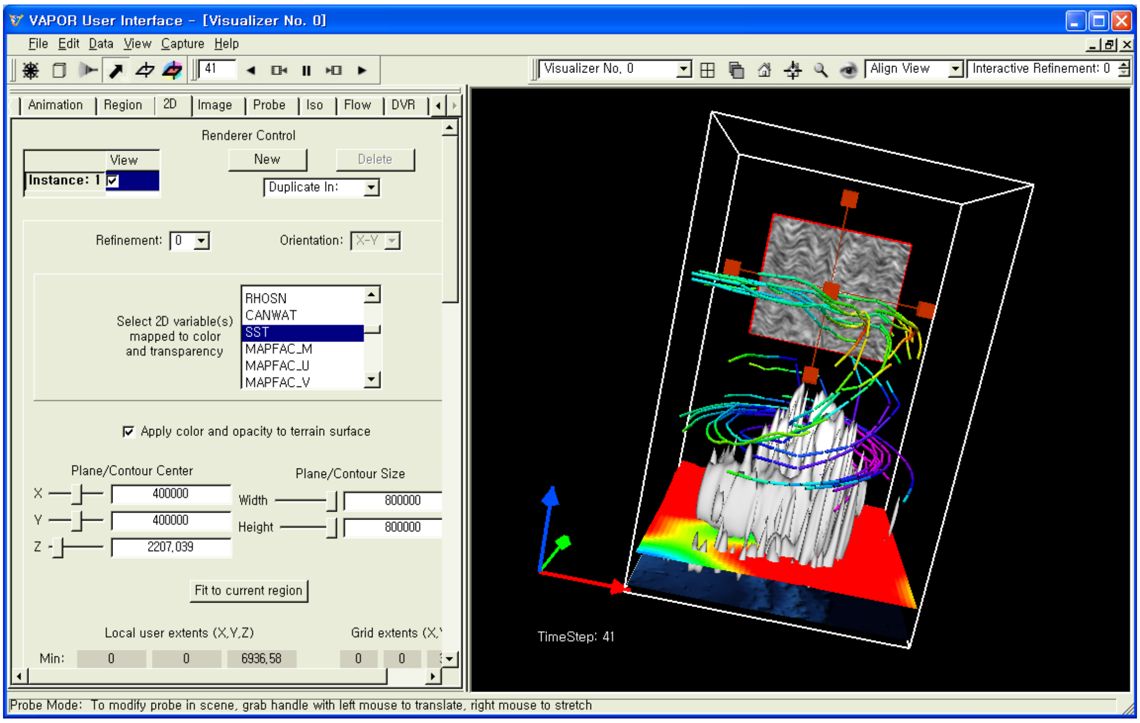


(c) Render instance 3: Visualization of 2D variable SST at time step=40

<Figure 5.4> Individual visualization of different variables using three 2D renderer instances in the same visualizer



<Figure 5.5> Visualizations of multiple instances using the three 2D renderer instances in the same visualizer



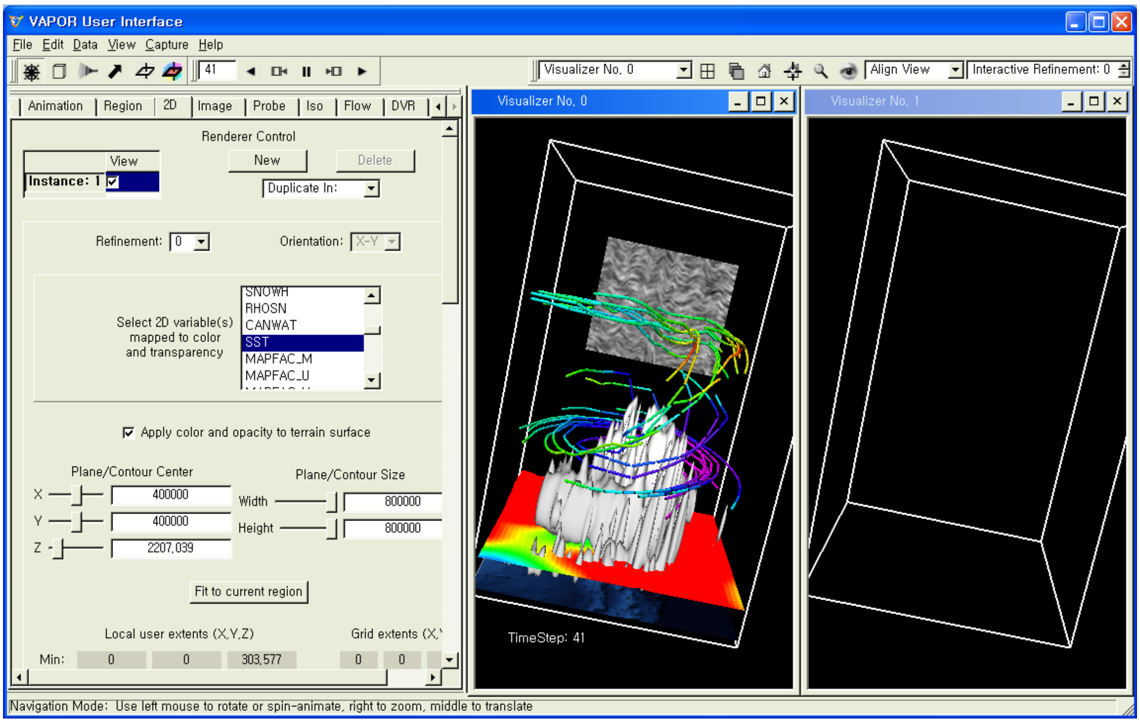
<Figure 5.6> Visualizations of multiple variables using various renderers (2D, Image, Probe, Iso, Flow) in the same visualizer

Note that only one "**DVR**" render can be enabled at one time in one visualizer, because of limitations in hardware support for volume rendering. However, any number of "**2D**", "**Image**" "**Probe**", "**Iso**", and "**Flow**" renderers can be enabled at the same time in the same visualizer window.

5.1.2 Multiple Visualizers

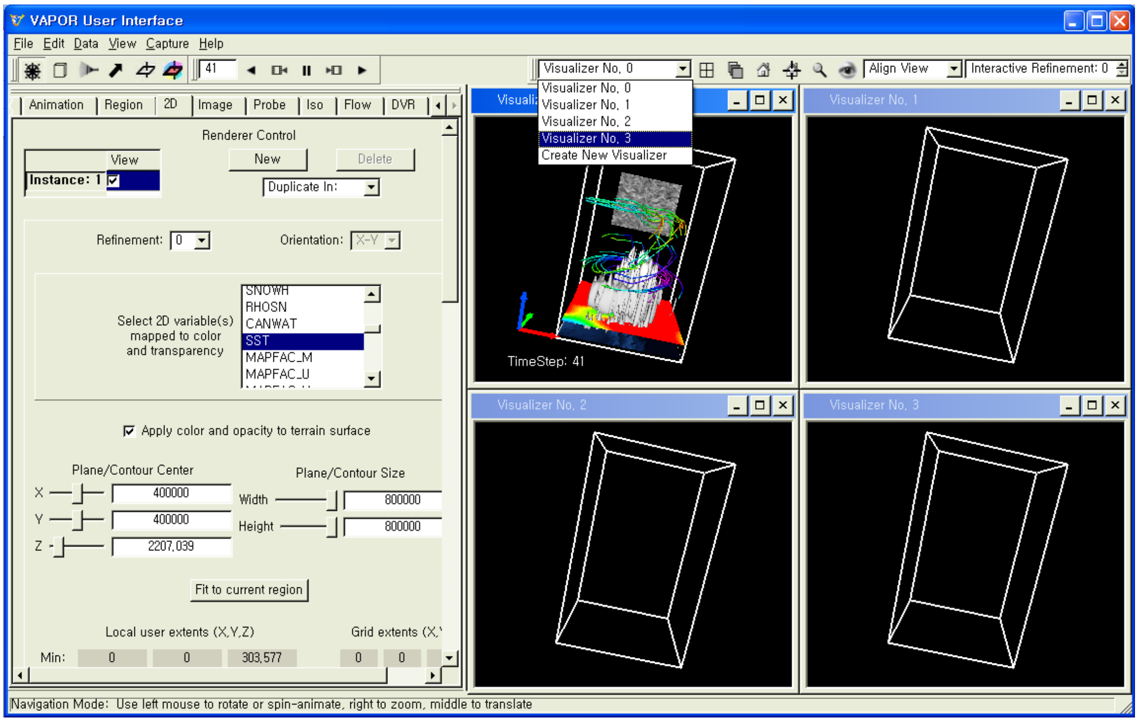
Sometimes it is useful to do side-by-side comparisons of different visualization results. Also, it is useful to visualize multiple variables at the same time in the different visualizer windows individually. Make your VAPOR GUI Window wider by grabbing the right edge with the left mouse and dragging it to the right, if your window is not already wide enough to show multiple visualizers.

Click the "Create New Visualizer" in the "Visualizer" selector on top of the visualizer window. You will see an empty window on the right of "Visualizer No. 0", labeled "Visualizer No. 1" like the following:



<Figure 5.7> Creating a new visualizer

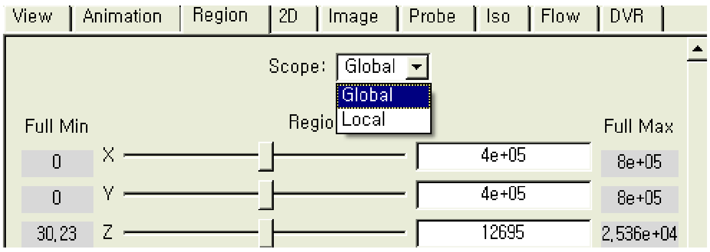
If you continue to select "Create New Visualizer" three times, you can see three more empty visualizers in the visualizer window like the following:



<Figure 5.8> Multiple visualizers

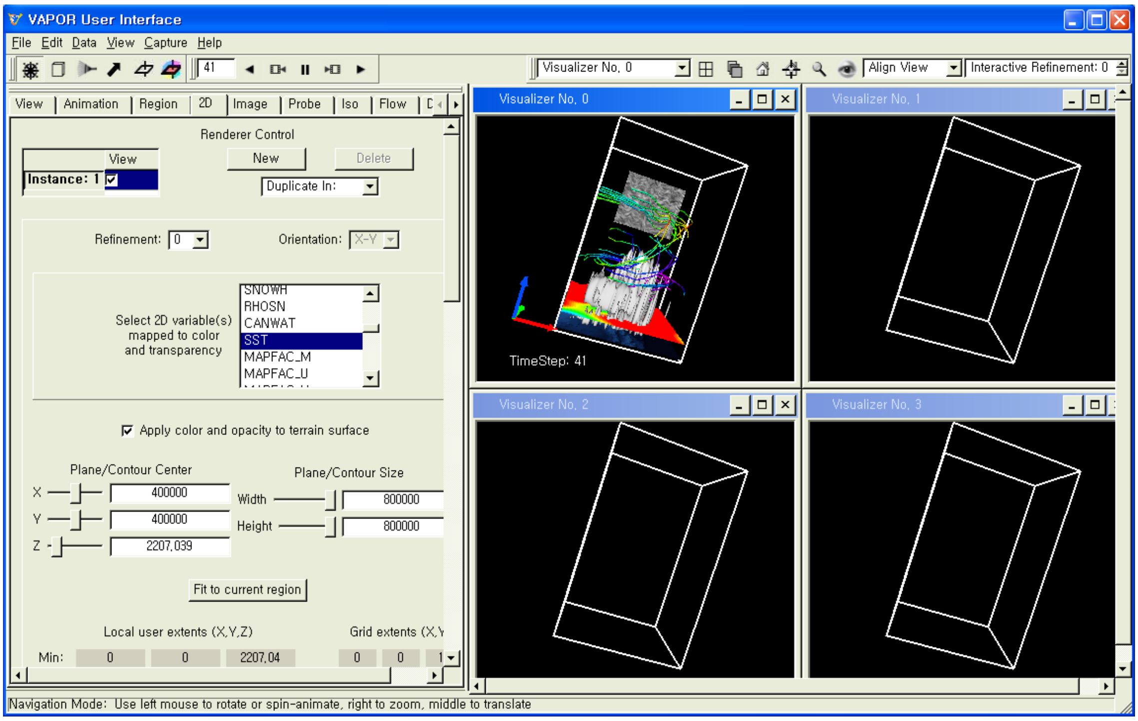
In this side-by-side visualization (or multiple visualization), some of the parameters in the tabbed panels may be shared among the visualizers, and others may not be shared.

In the “View”, "Animation" and "Region" render panels, there is a "Scope" selector indicating either "Global" or "Local" scope on top of each panel. By default, "Scope" is set to "Global".



<Figure 5.9> Scope options in the "Region" panel

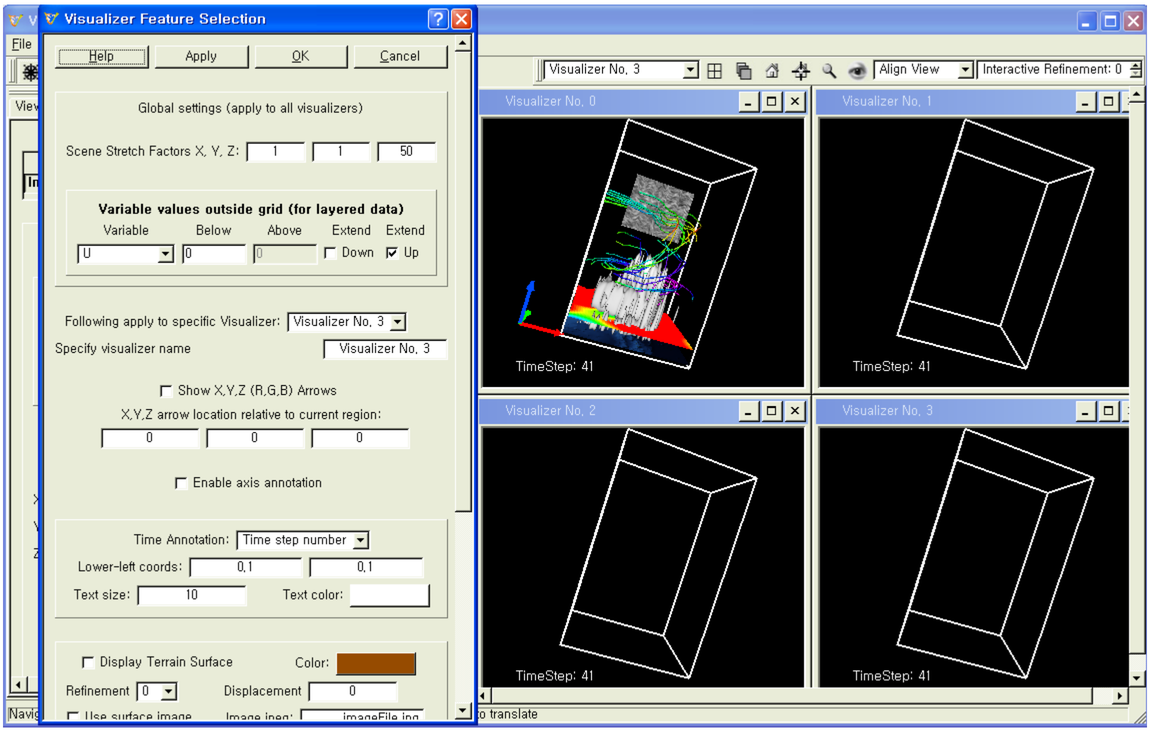
If you click and drag the left mouse button in one of the visualizers, all scenes in the visualizer windows will rotate together because they are using the "Global" viewpoint parameters. Note that "Global" parameters are not available in renderer panels such as "**2D**", "**Image**" "**Probe**", "**Flow**", "**Iso**", and "**DVR**".



<Figure 5.10> Rotating scenes using a viewpoint with global scope

Note that parameter settings in the "Edit Visualizer Features" panel are not shared among the visualizers. Therefore, you have to set visualizer features (i.e. "Show X, Y, Z Arrows" or "Time Annotation" or "Display Terrain Surface" or "Use surface image", etc.) individually in each visualizer.

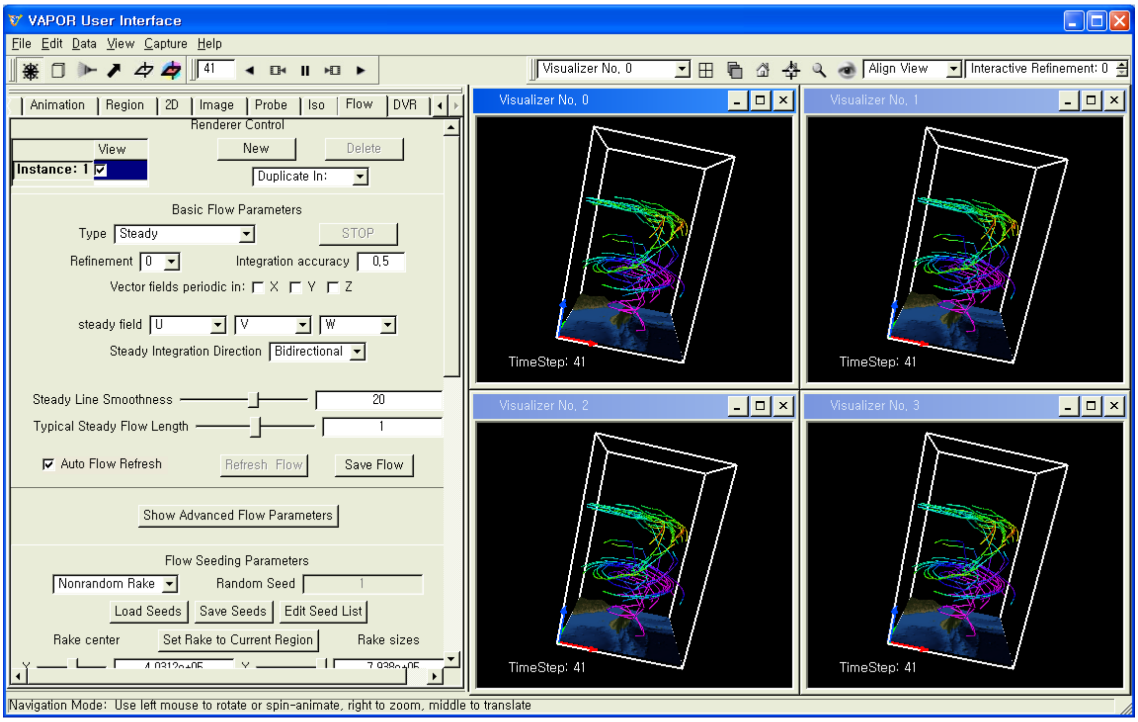
Activate the "Visualizer No. 1" visualizer by clicking in that visualizer window. Click on "Edit" --> "Edit Visualizer Features". Click in the check-box labeled "Show X,Y,Z Arrows". Select "Time step number" in the "Time Annotation" selector. Click "OK". Repeat the same processes in the "Visualizer No. 2" and "Visualizer No. 3" visualizers, respectively.



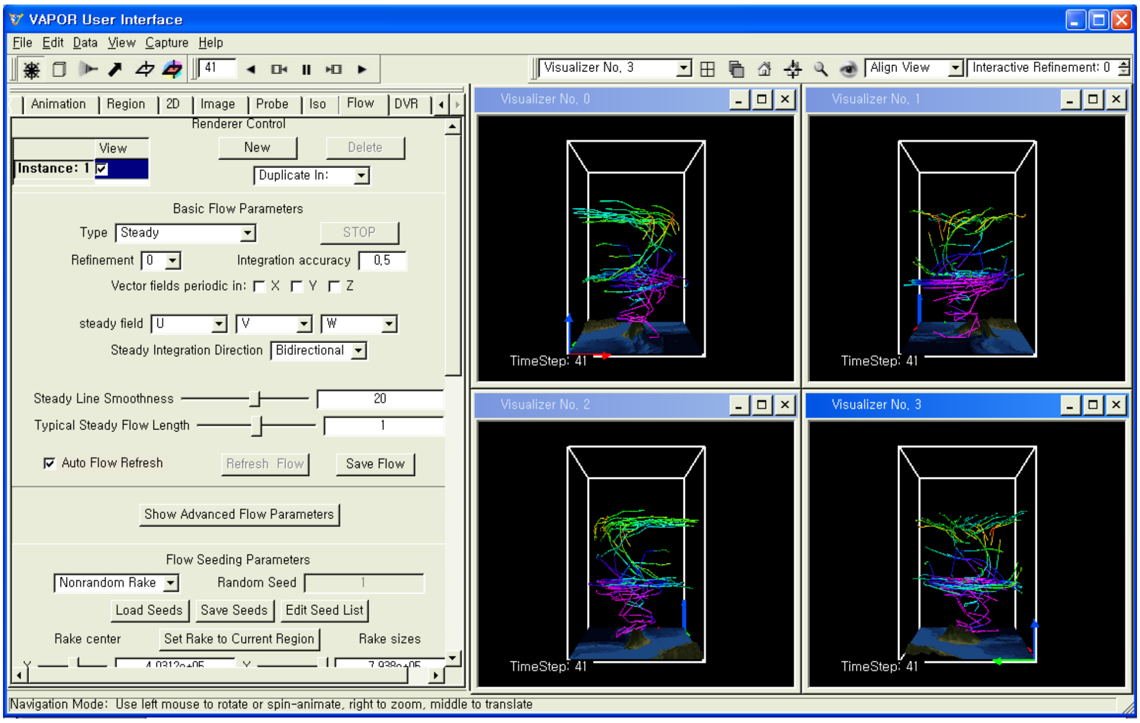
<Figure 5.11> Editing visualizer features in multiple visualizers

It is sometimes useful to see the same flow lines (or other visualizers) with different viewpoint. (Note: Instructions for setting up flow visualization are provided in section 5.3).

The flow rendering in the first visualizer can be replicated by copying the flow instance from "Visualizer No. 0" to all other visualizers, as in the following: Activate the "Visualizer No. 0" visualizer by clicking on it. On the "Renderer Control" at the top of the "**Flow**" renderer panel, click "Duplicate in:" and select "Visualizer No. 1", "Visualizer No. 2", and "Visualizer No. 3". Then activate the "Visualizer No. 1" visualizer, select the first "**Flow**" instance labeled "Instance: 1" and delete it (click "Delete" button). The first instance is the default one that was there when you created the visualizer window. Finally click in the check-box to enable the remaining flow instance labeled "Instance: 2". Repeat the same processes in the "Visualizer No. 2" and "Visualizer No. 3" visualizers. For each visualizer, make that visualizer active by clicking in the visualizer window. Then, click the "**View**" tab, and select the "Scope" to be "Local". The viewpoint in each visualizer will revert to the default viewpoint. Set the new viewpoint you want in each visualizer. You can now rotate the various scenes separately.



(a) Default global viewpoint in all four visualizers



(b) Different viewpoints looking at the same flow rendering

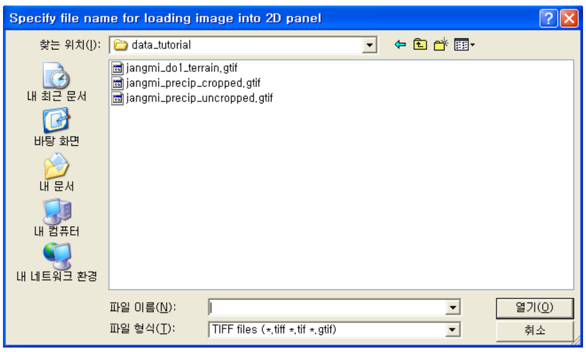
<Figure 5.12> Viewing the same flow lines in different viewpoints

5.2 Visualizing georeferenced images

VAPOR supports loading a georeferenced image into the visualizer and applying the image to the terrain. A georeferenced image contains information, either within itself, or in a supplementary file (a world file), that explains to a GIS system, how to align the image with other data. Formats that support built-in georeferencing include geotiff, jp2, and MrSid. Any horizontally oriented tiff image can be georeferenced, by providing the Tiff tags that specify its world position via a map projection. VAPOR can use the georeferencing information in geotiffs to insert these images into the correct place in a 3D scene. VAPOR supports georeferenced images using the following map projections (also supported by WRF-ARW): Lambert conformal conic, Mercator, latitude/longitude and polar stereographic.

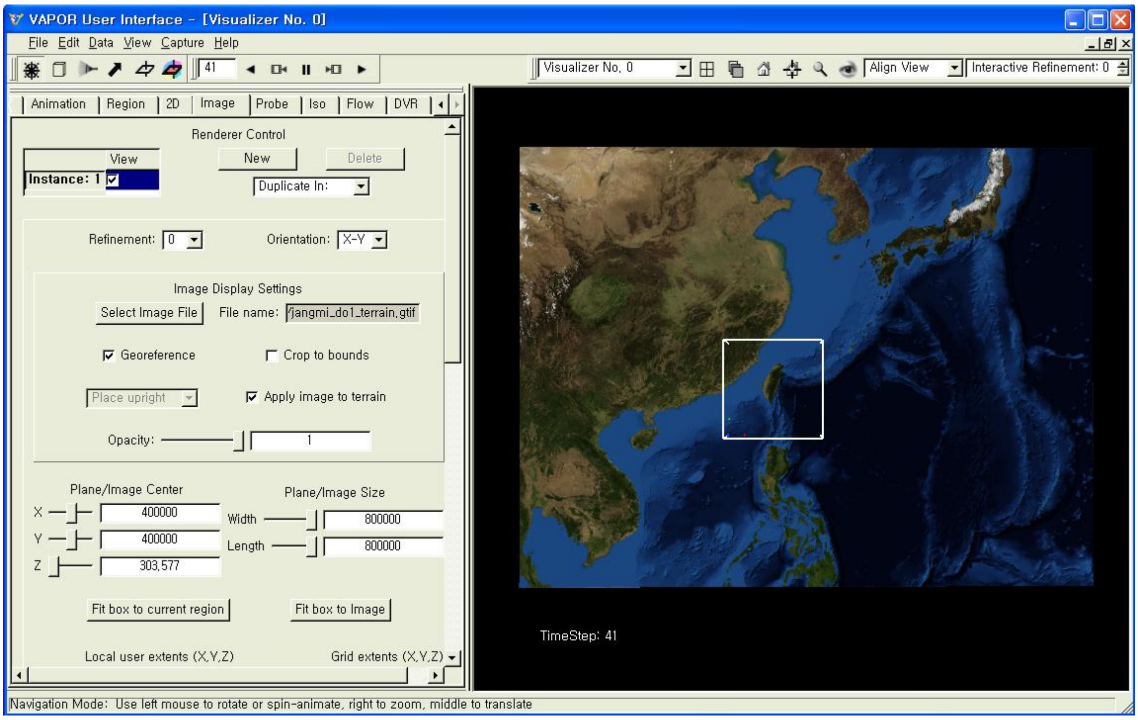
In the VAPOR "**Image**" panel, users can load a georeferenced satellite image and georeference 2D images created by other visualization software such as NCL. The method of converting a tiff image file into a geotiff (georeferenced image file) is described in detail in Chapter 3 and section 2 of the VAPOR/WRF Data and Image Preparation Guide.

Click "**Image**" tab. Make two new image renderer instances. For each image render, click "Select Image File". From the "Specify file name for loading image" dialogs, select one geotiff file for each instance.
("*jangmiTerrain.tiff*", "*jangmi_precip_cropped.tiff*", "*jangmi_precip_uncropped.tiff*").

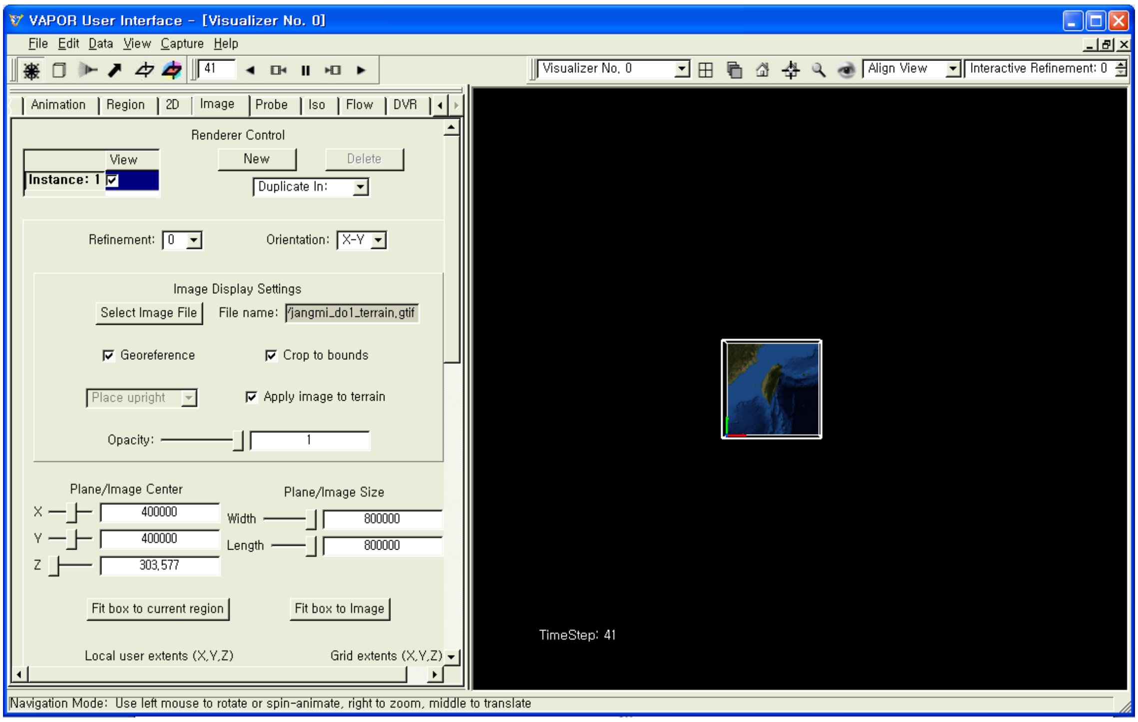


<Figure 5.13> Loading image dialog

Make sure the check box labeled "Georeference" is checked. Note that if you don't check it (or un-check it), the loaded satellite image is will be located within the box (not shown). If you click the check box labeled "Crop to bounds", portions of the image outside of the box will be removed, as in the following figure:

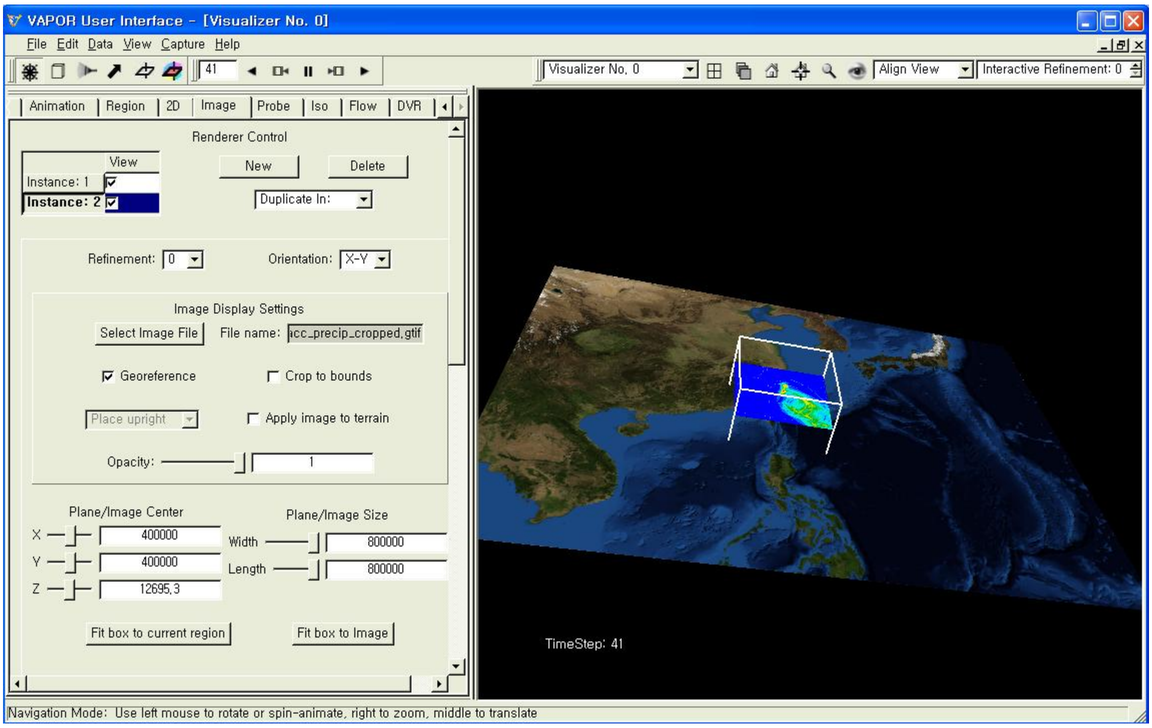


(a) Before checking Crop to bounds

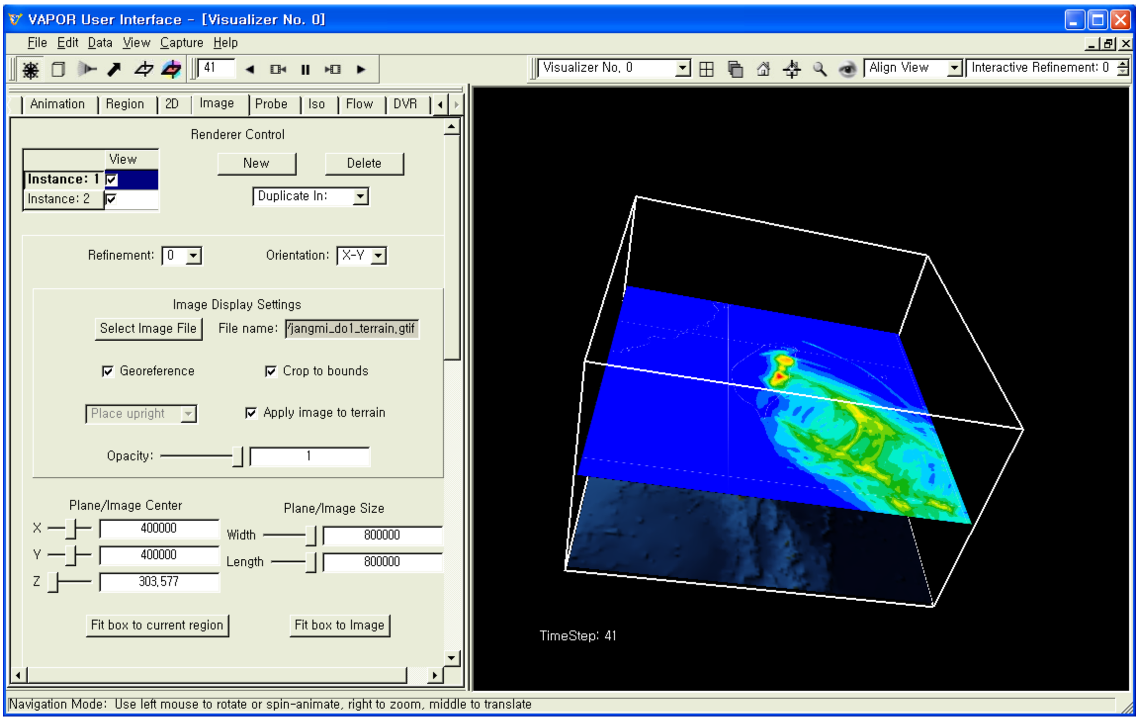


(b) After checking Crop to bounds

<Figure 5.14> Loading a georeferenced satellite image



(a) Before checking Crop to bounds



(b)After checking Crop to bounds (on the satellite image) and zooming in

<Figure 5.15> Loading a georeferenced 2D image

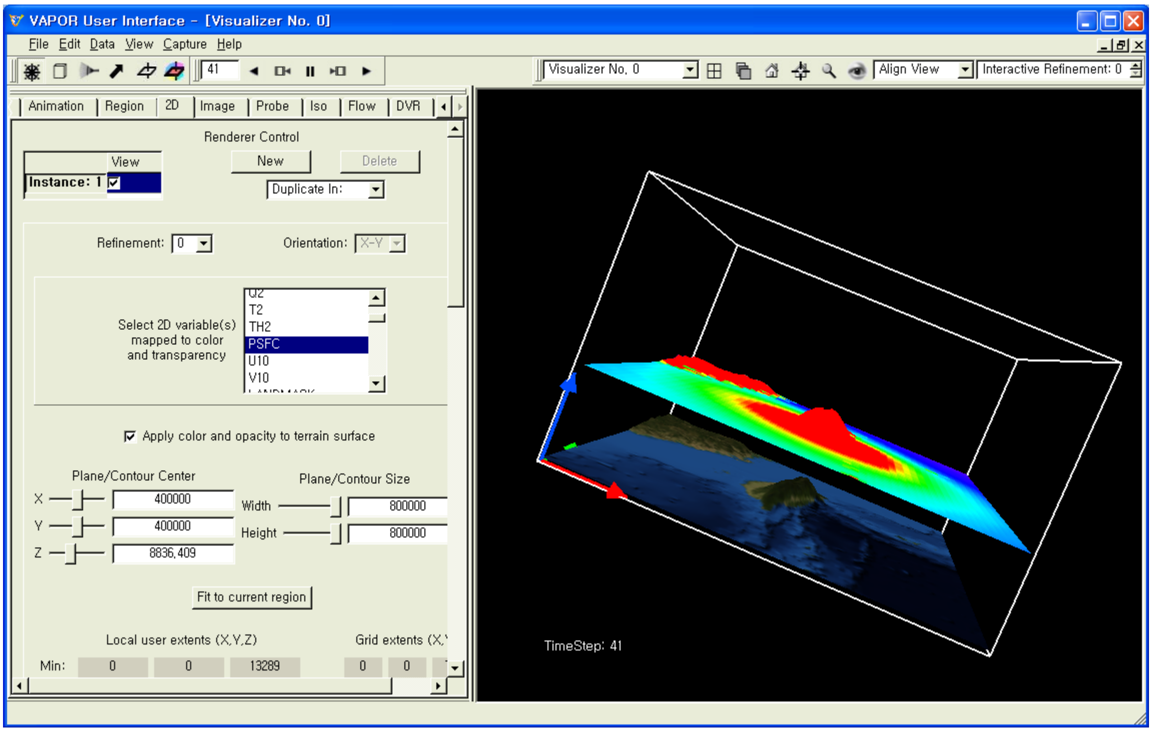
5.3 2D Visualization

5.3.1 Constructing Contours in 2D Plane

Note that 2D variables in a dataset must be horizontally oriented (a function of the X and Y coordinates in the visualizer) to be visualized using VAPOR. Currently, all the 2D variables in a WRF dataset are horizontal. 2D variables can be displayed on a horizontal plane or can be applied to the terrain in the visualizer.

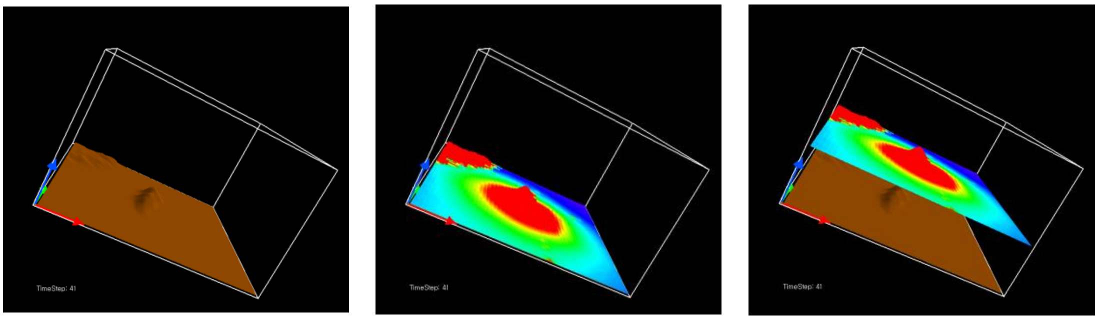
The "jangmi_lowres_data" directory, converted from the WRF model's simulation data of Typhoon Jangmi to VAPOR data, includes several 2D and 3D variables. Click the "**2D**" tab. In the "2D Variable(s)" selection box, you will see a list of 2D variables in the dataset.

Scroll down the "2D Variable(s)" selection box to see the "PSFC" variable name and select it. Then click the check-box of "Instance: 1" at the top of the "2D" render panel.



<Figure 5.16> 2D image of PSFC at time step=41

To provide a more useful visualization of 2D variables we can use 2D variables to color the terrain surface. In the "**2D**" render panel, select "PSFC" variable, and type the value "100" in the "Z" coordinate box under "Plane/Contour center", and then check the check-box labeled "Apply color and opacity to terrain surface". This causes the color to be applied to a surface that is vertically displaced 100m above the ground, ensuring that the 2D plane will appear above the existing surface image.

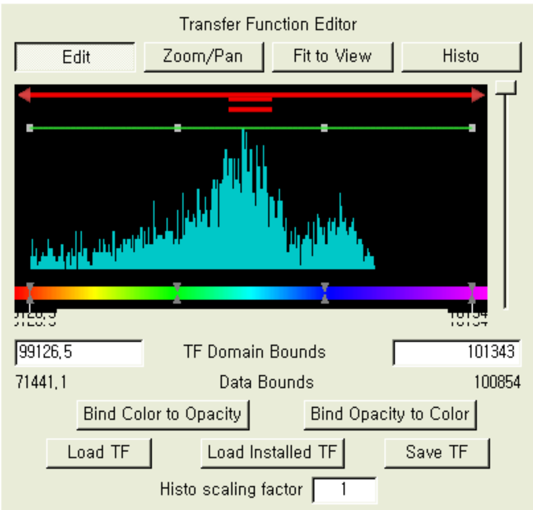


(a) model terrain (b) at z=100 (c) at z=10000

<Figure 5.17> 2D images applied to a model terrain surface

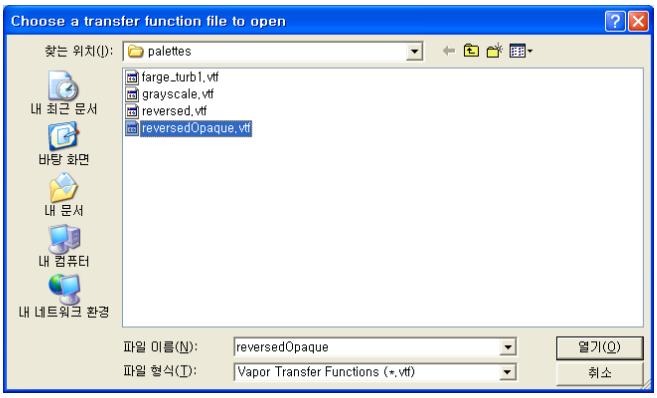
5.3.2 Editing the Transfer Function

In the "**2D**" render panel, scroll down to the bottom of the panel where a "Transfer Function Editor" is provided.



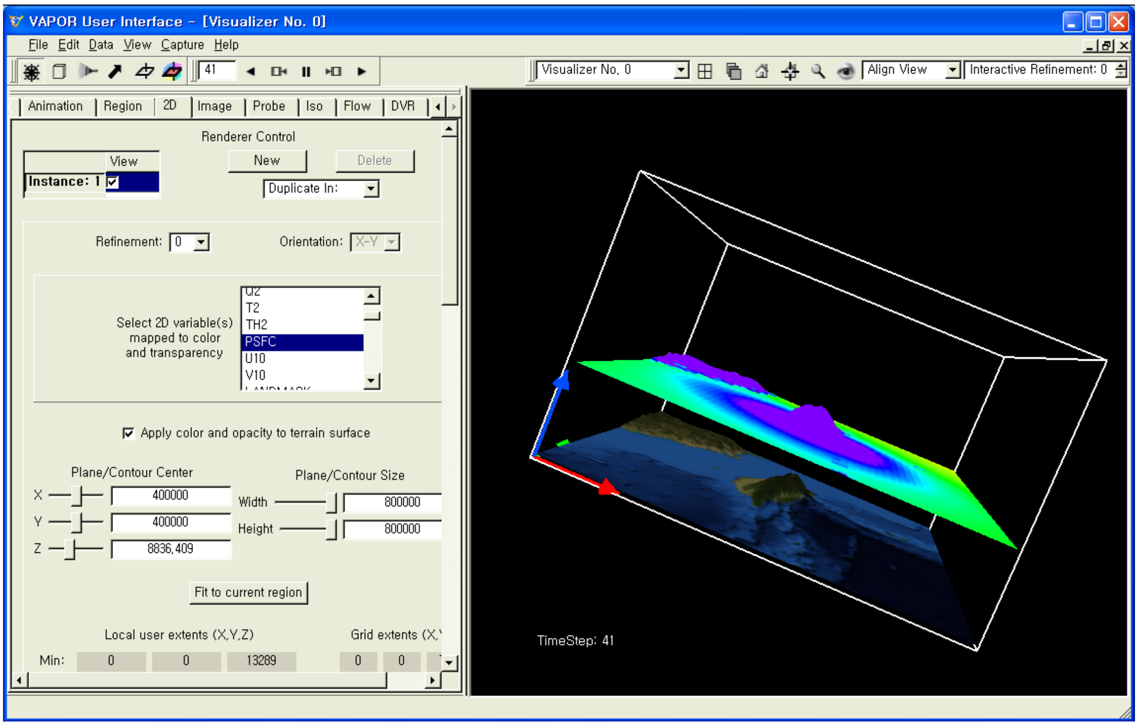
<Figure 5.18> Transfer Function Editor

You can change the colors in the 2D images. Click the button labeled "Load Installed TF". In the file selection dialog that pops up, select the file "reversedOpaque.vtf", and click "Open". The resulting transfer function has constant full opacity, with the color mapping reversed.



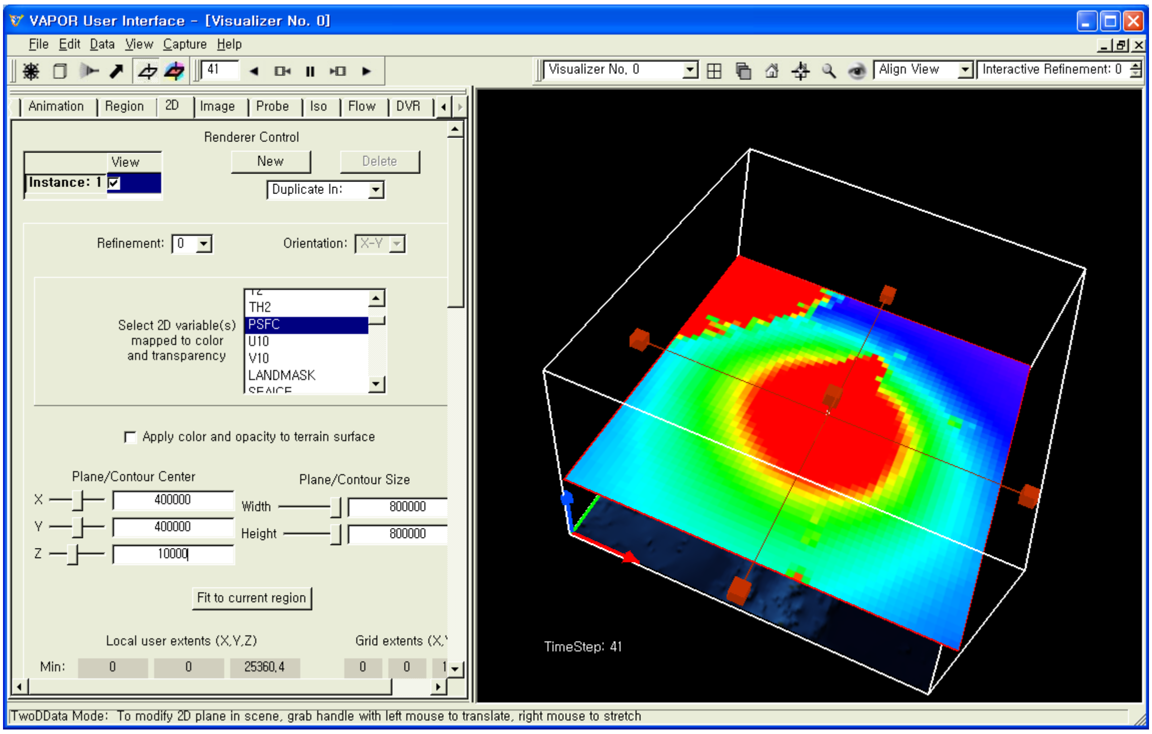
<Figure 5.19> Transfer function file selection dialog

If you select "reversedOpaque.vtf" file, and increate the Z coordinate of "Plane/Contour Center", the colors of the 2D image in <Figure 5.16> will be changed like the following:

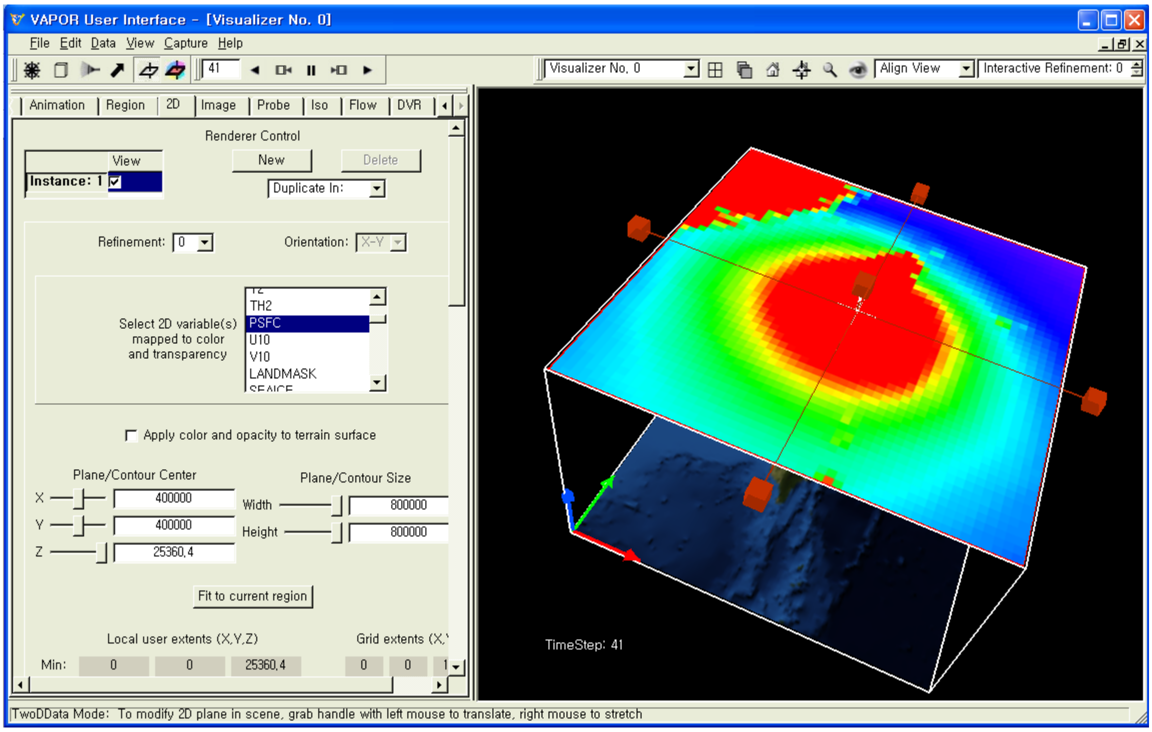


<Figure 5.20> Image of PFSC as in <Figure 5.16>, using a different transfer function

You can move the 2D plane vertically by clicking on the 2D Mode button and then dragging the 2D box handle (with the left mouse) like the following:



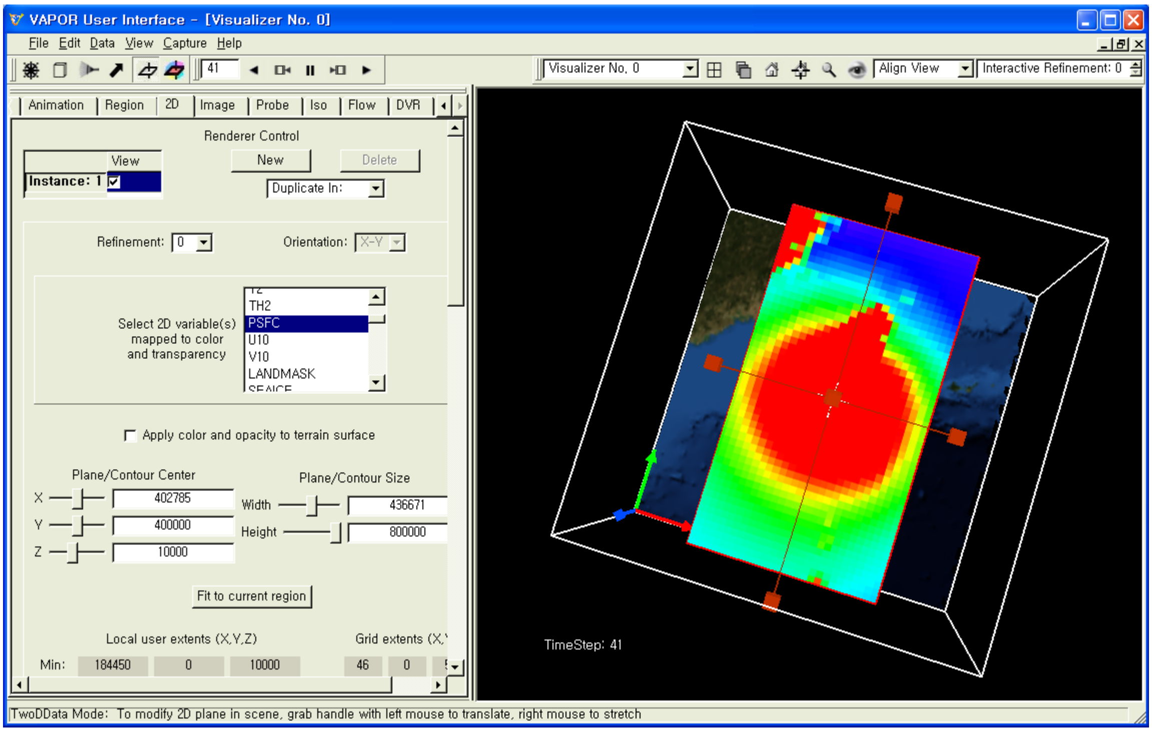
(a) moving to the height=10000m



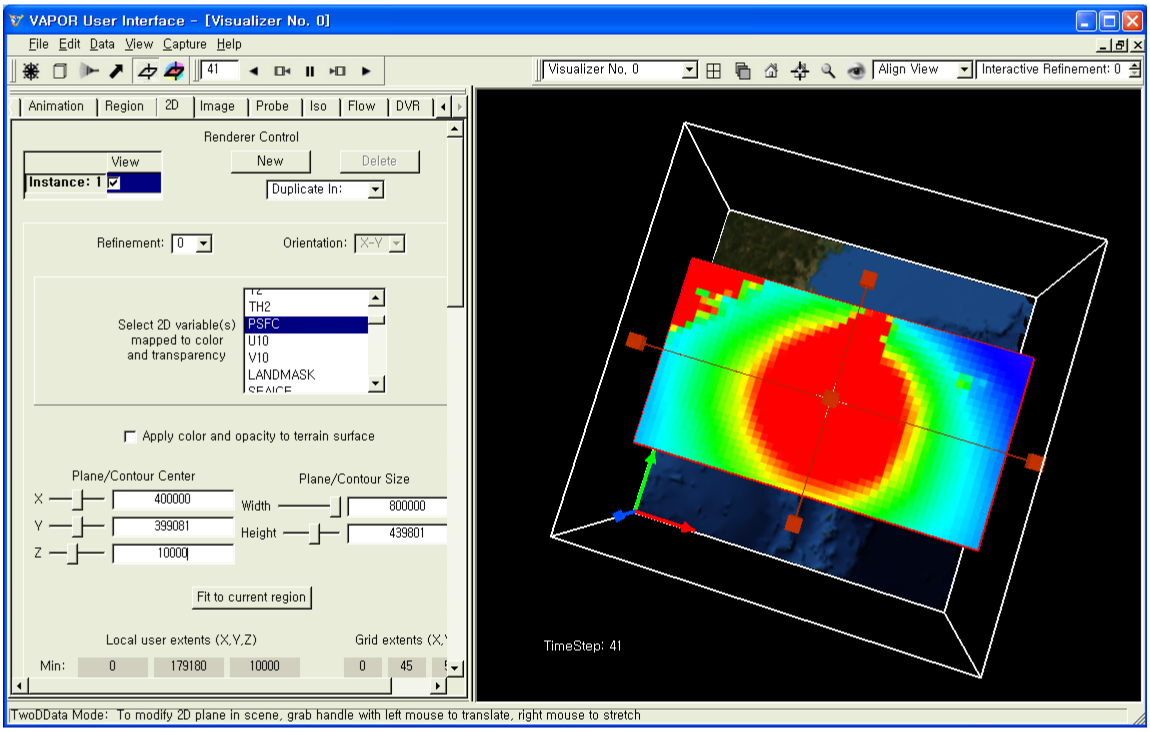
(b) moving the 2D plane to the height=25360.4m (top of the domain)

<Figure 5.21> Moving the 2D plane vertically in the domain

You can resize the 2D plane horizontally by grabbing the box edge handle with the right mouse and dragging it like the following:



(a) resizing in X-direction



(b) resizing in Y-direction

<Figure 5.22> Resizing the 2D plane horizontally in the domain

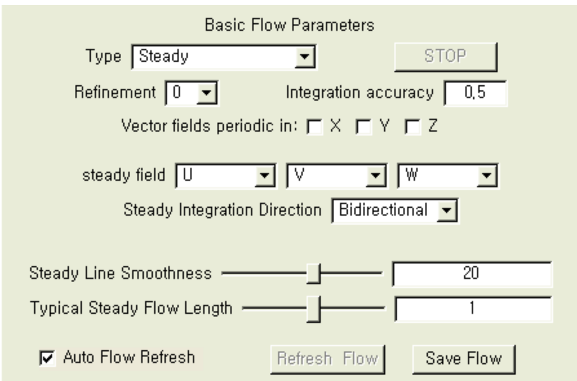
5.4 Flow Visualization (Streamlines and Trajectories)

VAPOR visualizes a flow line by performing a numerical integration of a vector field in a volume dataset. The numerical integration starts at user-specified starting points, called "seed points", and tracks the successive positions in the volume dataset that result from the integration. The successive positions can be visualized with various geometric entities.

5.4.1 Constructing random streamlines in the wind field

In the "**Region**" tab, click the button "Maximize Region in Full Domain", so that we will be able to integrate flow lines in the full data domain. Click the "**Flow**" tab. You need to establish some basic parameters before designing an interesting flow visualization.

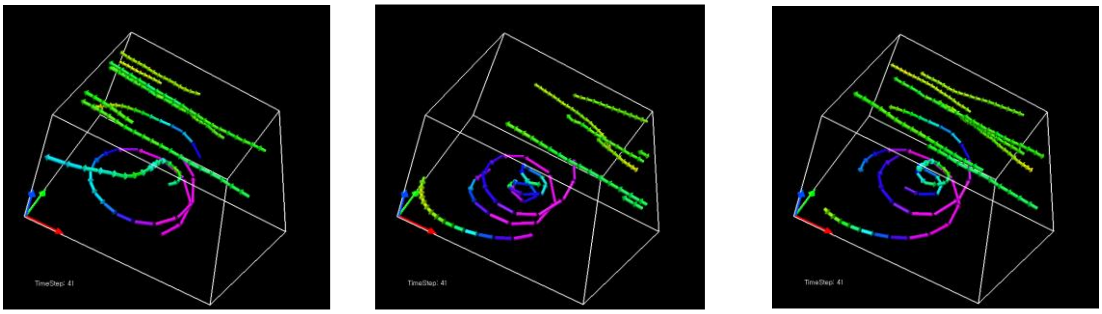
First, you must specify three variables that define the vector field to be visualized. The wind is defined by the three component (U, V, W), so set the X, Y, and Z-coordinate selectors (labeled as "steady field") to U, V, W (if they are not already set that way). Note that by default, the flow type is "Steady". Click the "Instance:1" checkbox to see the flow lines resulting from the current settings.



<Figure 5.23> Basic flow parameters in the "Flow" renderer panel

"Steady" means that the vector field timestep does not change during the flow integration. The successive positions of a flow line in a steady flow depend only on the coordinates along the flow line at the current time, not on the elapsed time.

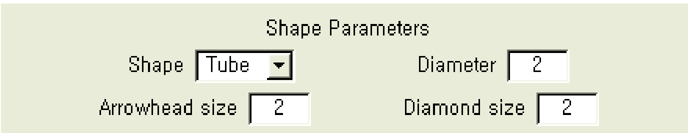
You can select whether flow integration goes "forward", "backward", or in both directions from the seed points. By default, it is set to "bidirectional".



(a) forward (b) backward (c) bidirectional

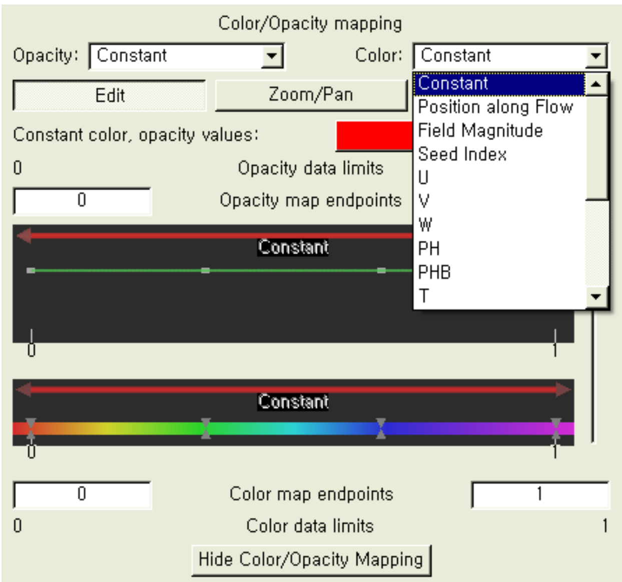
<Figure 5.24> Comparison of flow integration directions

You can set the flow shape to "Tube", "Point", or "Arrow". By default, it is set to "Tube".



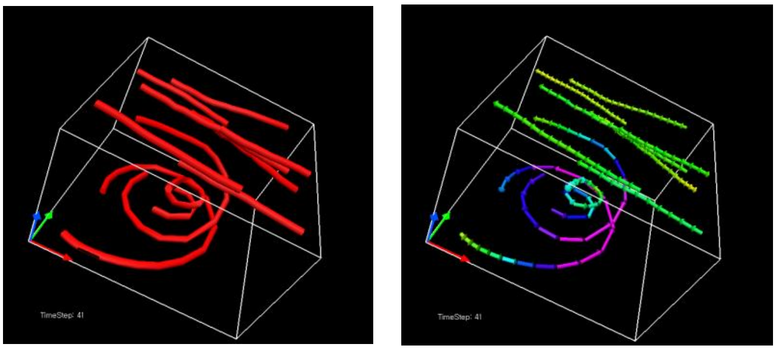
<Figure 5.25> Flow shape parameters

You can set the flow color to "Constant", "Position along Flow", "Field Magnitude", "Seed Index", or to any 3D variable in the volume dataset.



<Figure 5.26> Flow color/opacity mapping parameters

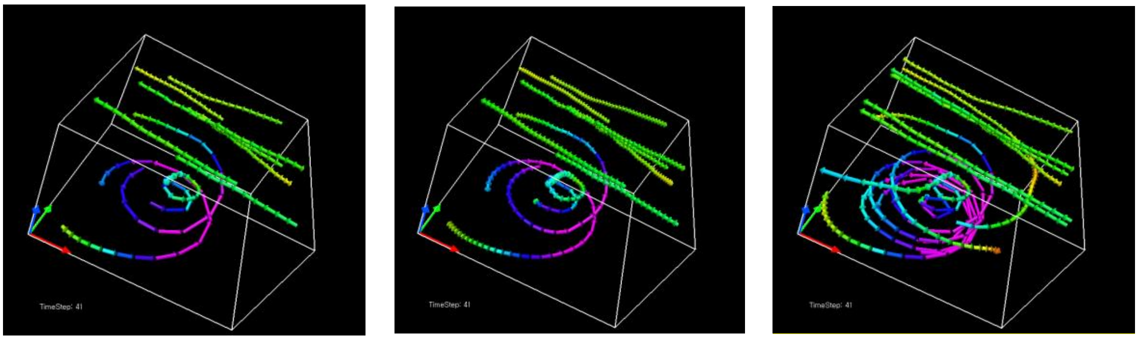
Set the flow "Shape" to "Arrow", change the value of "Diameter" to "1", and set the flow "Color" to "Field Magnitude". Make sure your current region is the full domain. You should see the modified flow images like the following:



(a) default (b) modified

<Figure 5.27> Comparison of flow shape and diameter

You can choose how long or short you want the flow lines to be, and how smooth they should appear. Slide the "Steady Line Smoothness" slider to the right to a value of about 40, and set the "Typical Steady Flow Length" to about 2 (or type values in the box next to the slider directly). These values are based on the average magnitude of the field, and will result in flow lines that are of length about 2 times the volume diameter, and each line having up to 400 sample points along its length.

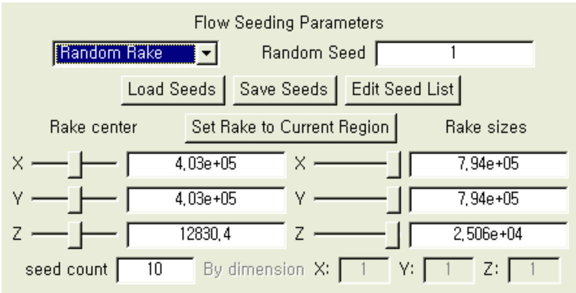


line smoothness=20 flow length=1	line smoothness=40 flow length=1	line smoothness=20 flow length=4
(a) default	(b) modified length	(c) modified smoothness

<Figure 5.28> Comparison of flow line smoothness and flow length

Second, you need to establish the seed points for the flow, i.e. the points that will be used to start the flow integration. Scroll down the section entitled "Flow Seeding

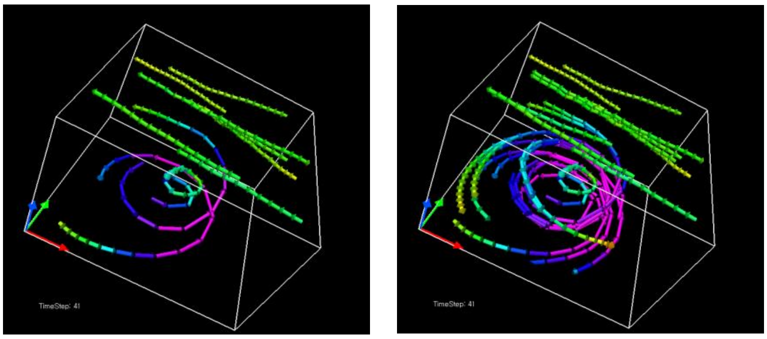
Parameters". Note that the seed points are initially set to use a "Random Rake". This means that the seed points are randomly generated within the bounds of the box-shaped region defined by the "Rake center" and "Rake sizes". By default, seed count is 10.



<Figure 5.29> Flow seeding parameters

Click the button "Set Rake to Current Region". (If a seed point is not inside the current region, there will be no flow line generated.) Set the "seed count" to 10 or 20, so that you can see 10 or 20 flow lines resulting from the random seed points. This will give us a coarse idea of the wind flow.

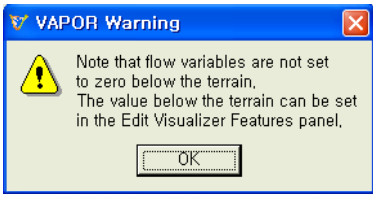
The streamlines are by default a constant red color. To give an indication of how strongly the wind is blowing, you can color them according to the wind speed. Set the flow color to "Field Magnitude".



(a) seed count=10 (b) seed count=20

<Figure 5.30> Comparison of number of seed points

There may be a warning message indicating that the flow variables (U, V, W) are not zero at the bottom of the data like the following:



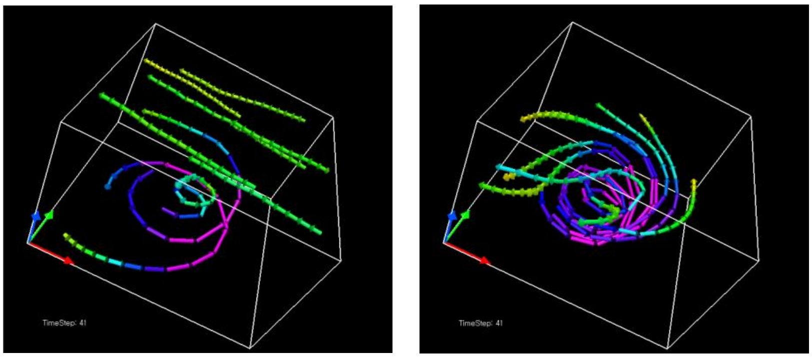
<Figure 5.31> Flow warning message

This can have the undesirable effect of causing wind to appear to flow through and below the terrain. To fix this issue, click on "Edit" --> "Edit Visualizer Features". Near the top of the "Edit Visualizer Features" panel, there is a frame labeled "Variable values outside grid (for layered data)". Select the variable "U", uncheck the check-box labeled "Extended Down", and specify the value of 0 in the box labeled "Below". Do the same for the variables "V" and "W". Now the wind field will be always equal to zero below the terrain. Click "OK". See <Figure 4.17>.

5.4.2 Constructing uniformly placed streamlines in the wind field

You can visualize the wind direction by using uniformly placed seed points instead of randomly placed seed points in previous section. Let's construct a 3 by 3 array of seed points, placed in a rectangle slightly above the ground.

Return to the "**Flow**" renderer panel and scroll down to the "Flow Seeding Parameters". Select "Nonrandom Rake" instead of "Random Rake" (Default). At the bottom of the "Flow Seeding Parameters", type in values 3, 3, 1 into the three text-boxes next to "By dimension X:... Y:... Z:...". This will result in a uniformly spaced array of seed points, of sizes 3x3x1 (in the x, y, and z-directions), centered within the bounds of the current rake (which is also the current region).

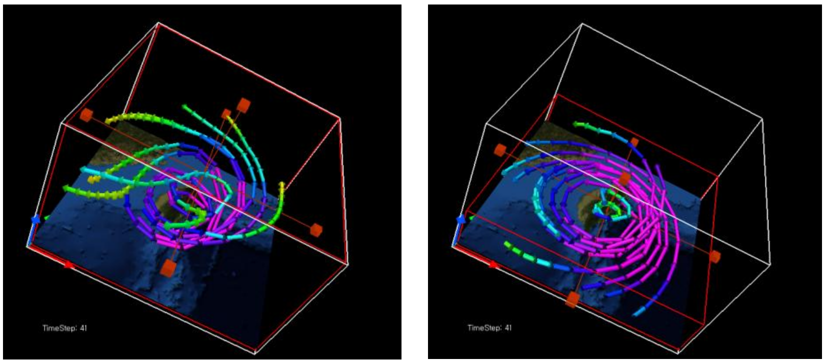


seed count=9　　　　　　seed count=9

(a) random rake　　　　　　(b) nonrandom rake

<Figure 5.32> Comparison of "Random Rake" and "Nonrandom Rake"

In order to position the wind arrows near the ground, click the rake icon near the upper left corner of the VAPOR GUI Window. You will see a red box with handles, indicating the extents of the rake. Grab the top handle of the rake with the right mouse button, and slide it down to where the top of the box is 1/10 of the height of the region. The flow arrows will now start close to the ground, similar to the following:



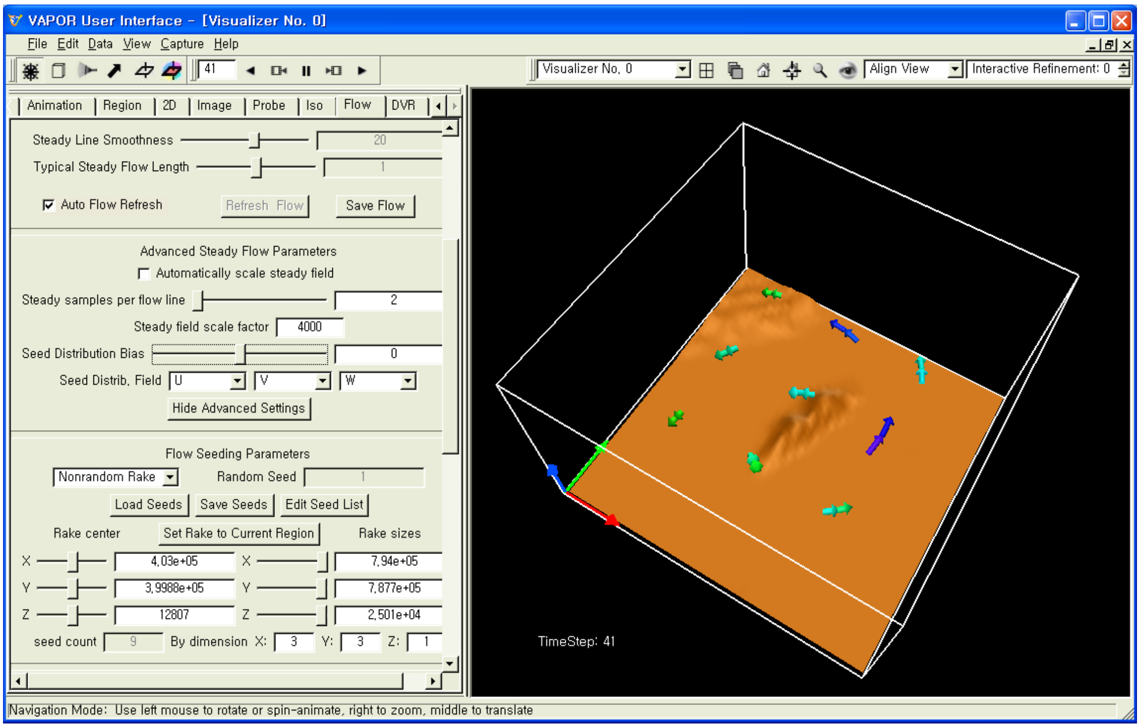
(a) default　　　　　　(b) modified

<Figure 5.33> Positioning wind arrows

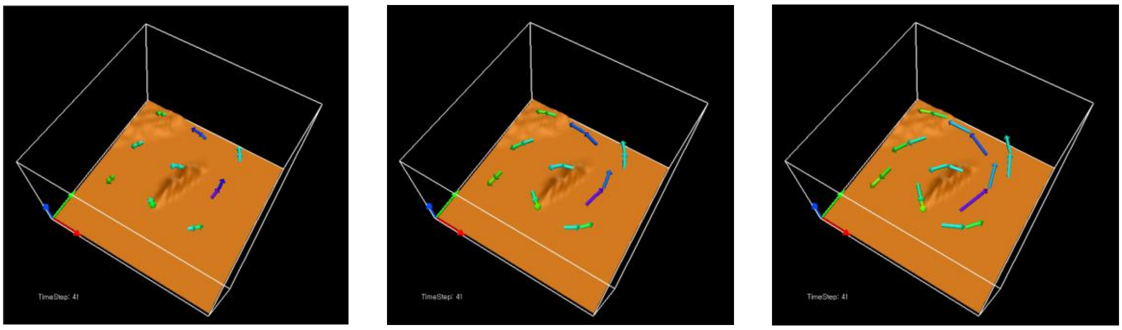
5.4.3 Constructing wind arrows

Instead of drawing streamlines, you can construct wind arrows, indicating direction and velocity at each seed point. This requires disabling the automatic calculation of flow length so that we can use a single arrow at each seed point.

In the "Basic Flow Parameters", set the "Steady Integration Direction" to "Forward". Click the button "Show Advanced Flow Parameters". Uncheck the check-box labeled "Automatically scale steady field". Set the "Steady samples per flow line" to the minimum (2), and change the "Steady field scale factor" to 4000, or so. You will see 3x3 array of arrows pointing in the wind direction, of length proportional to the wind speed, like the following:



<Figure 5.34> Uniformly spaced arrows indicating wind direction



(a) scale factor=4000 (b) scale factor=8000 (c) scale factor=12000

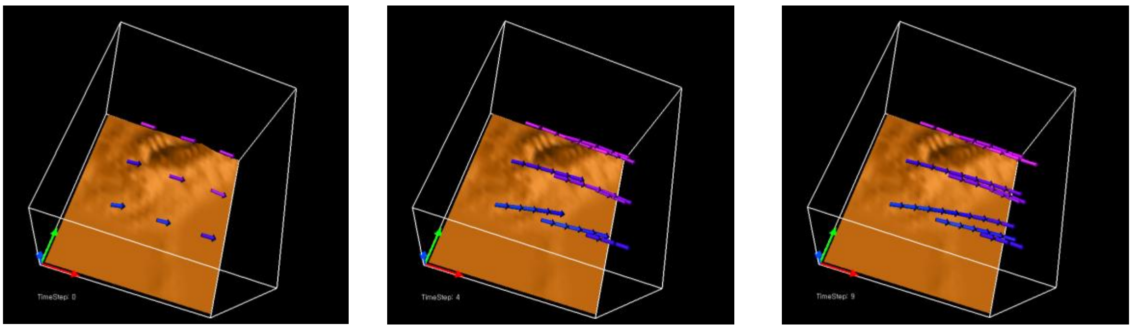
<Figure 5.35> Comparison of steady field scale factors

5.4.4 Constructing trajectories in the wind field

Unsteady flow visualization can be used to track the motion of wind currents over time, or to see particle trajectories. Specifying the flow requires establishing a set of seed points; however, with unsteady flow, the seed points at one time step determine the flow positions at subsequent time steps. Each seed point is moved by integrating the flow field for one time step, resulting in a new point at the next time step. Note that unsteady flow integration in VAPOR assumes that the domain is not moving; unsteady flow lines will be incorrect if integrated across moving domains. Unsteady flow trajectories are valid with the **april_lowres** dataset, but are not valid with the **jangmi_lowres** dataset, because the **jangmi_lowres** uses a domain that moves in time.

Return to the "**Flow**" renderer panel. Disable the flow integration by un-checking the "Instance:1" checkbox. Set the "Type" to "Unsteady". Set the "Unsteady Integration direction" to "Forward". (Note that "Unsteady field scale factor" should be set to "1", since the correct scaling factors were derived from the WRF output files when the VAPOR data was constructed.) Then enable the flow. It will take a few seconds to integrate the flow, because U, V, and W at all time steps need to be read from disk.

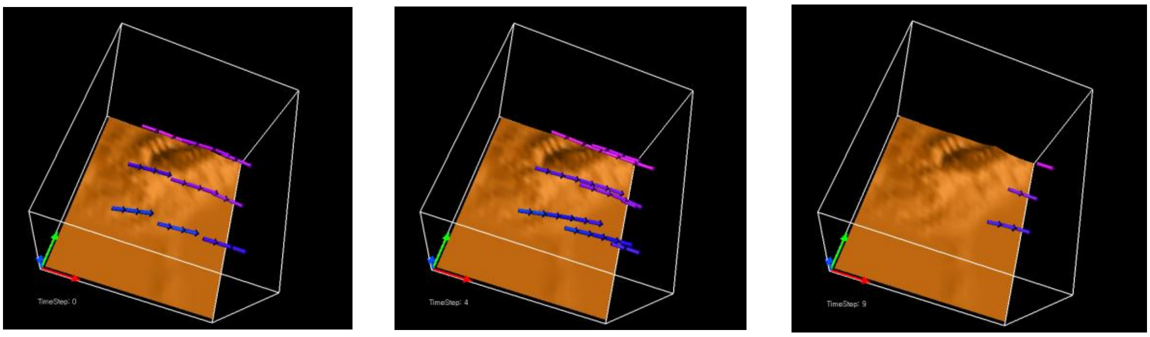
When the integration is complete, you will see that the 9 seed points have resulted in 9 path lines, describing the trajectories that would be taken by 9 particles.



(a) at time step=0 (b) at time step=4 (c) at time step=9

<Figure 5.36> Trajectories at display intervals (-100, 1)

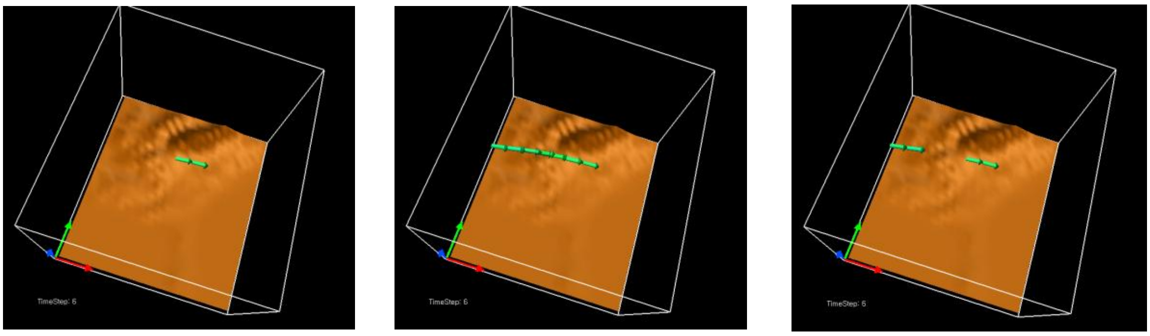
To animate the motion of these particles, scroll down to the "timestep display interval min, max" values at the bottom of the "Shape Parameters". Set these values to -3 and 3. This means that at each time step=t you will see the sequence of particle positions from time steps t-3 to t+3. You can animate this motion by clicking on the play button in the "Animation" panel.



(a) at time step=0 (b) at time step=4 (c) at time step=9

<Figure 5.37> Trajectories at display intervals (-3, 3)

To see the continuing motion of particles, set the "Seed time start, end, increment" values in the "Flow Seeding Parameters" section. These specify the first and last time steps that seeds are released, and the increment between subsequent seed release times. You can change the "Seed time start, end, increment" values as needed.



(a) start=0, end=0, increment=1 (b) start=0, end=20, increment=1 (c) start=0, end=40, increment=1

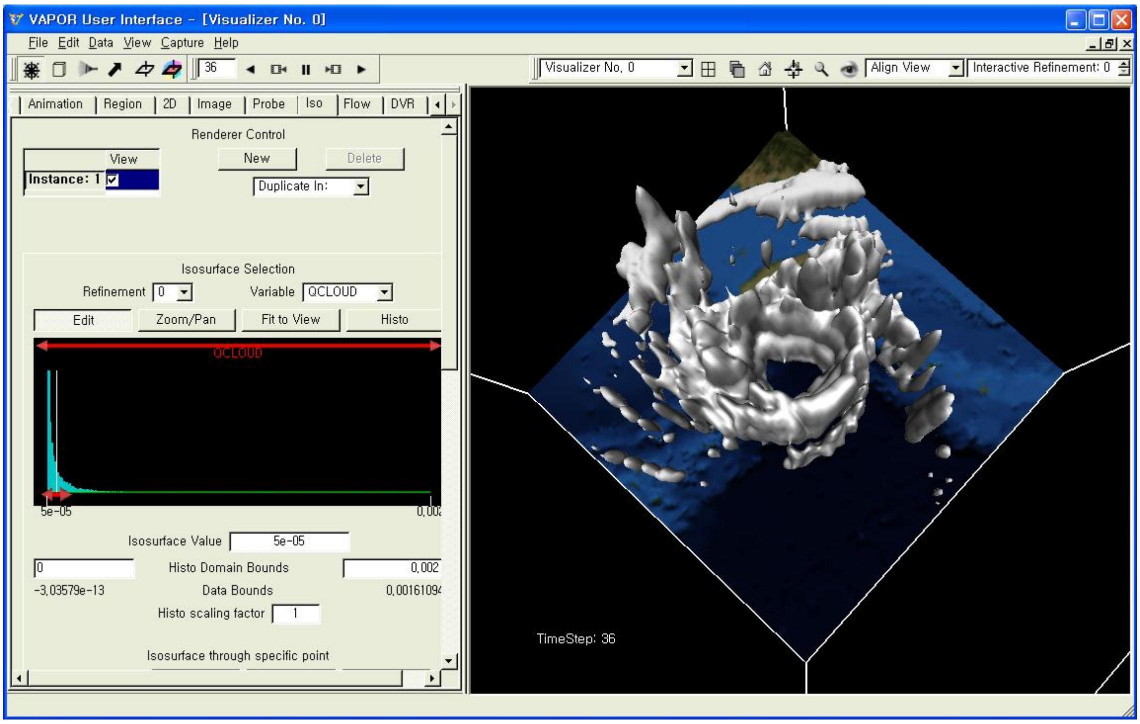
<Figure 5.38> Comparison of seed time start, end, increment at time step=6. Trajectories at display intervals (-1,1)

5.5 Isosurface Visualization

5.5.1 Constructing an isosurface

Disable the flow integration in the "**Flow**" panel. From the "Variable" selector in the "**Iso**" renderer panel, pick the variable "QCLOUD". Scroll down to the Isosurface Selection window (black), and click "Histo:" to see a histogram of the selected variable.

Below the histogram window, to the left and right of "Data Bounds", there are values -3.03579×10^{-13} and 0.00161094. These values indicate the min and max value of the variable "QCLOUD" at the (current) 36th time step. Type the values 0 and 0.02 in the left and right "Histo Domain Bounds" specifying an interval of values of "QCLOUD" that will be most interesting for observing the Typhoon. Type in an "Isosurface Value" of 0.00005. Press enter to set these values, then click the buttons "Fit to View" and "Histo" above the histogram window. Click in the check-box to enable an isosurface rendering.

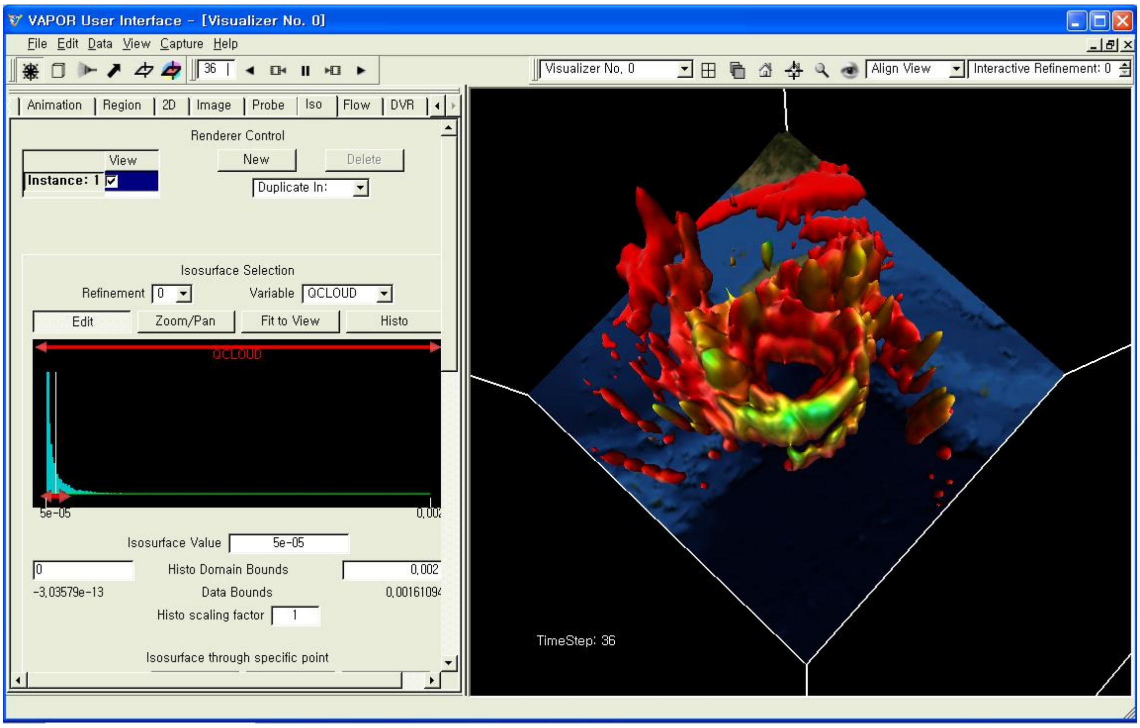


<Figure 5.39> Isosurface visualization with a value QCLOUD=0.00005

5.5.2 Mapping color onto an isosurface

VAPOR has the capability of coloring an isosurface based on the values of another variable in the dataset.

Disable the isosurface rendering by un-checking the "Instance: 1" check-box. Scroll down to the bottom of the "**Iso**" renderer panel. In the "Isosurface Appearance" section, using the "Mapped Variable" selector, pick the variable "W". There is also a section labeled "Transfer Function Editor" in which the variable "W" is now identified with red text. Type the values 0 and 5 in the left and right "TF Domain Bounds" specifying an interval of values of "W". Press enter to set these values, then click the buttons "Fit to View" and "Histo" above the "Transfer Function Editor" window. Click in the check-box to enable an isosurface rendering. You will see the isosurface of "QCLOUD" colored by the values of "W" as in the following:



<Figure 5.40> Isosurface visualization of QCLOUD colored by the values of W

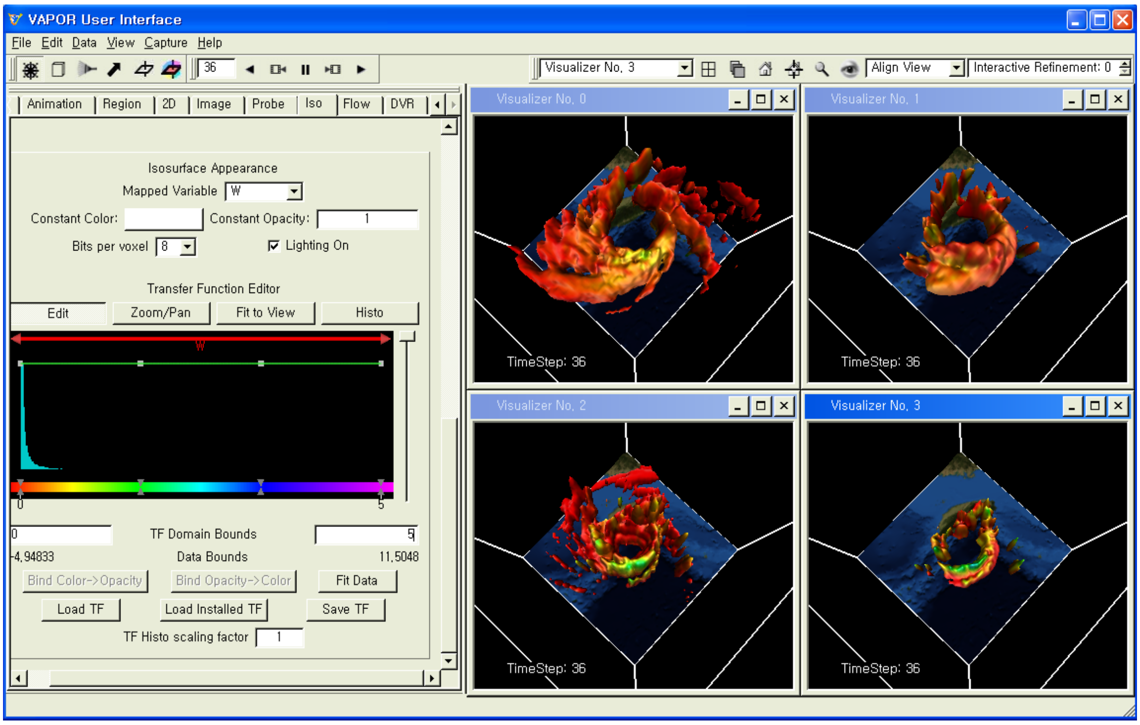
5.5.3 Constructing multiple isosurfaces

In typhoon research, multiple visualizations for the four variables (QICE, QSNOW, QCLOUD, QRAIN) are useful. Multiple visualizers can be created using the visualizer selector near the top of the visualizer window. (See section 5.1)

Click the "Create New Visualizer" three times in the visualizer selector. Set the variables "QICE", "QSNOW", "QCLOUD", and "QRAIN" in the "**Iso**" tab after clicking in Visualizer No. 0, No. 1, No. 2, and No. 3, respectively.

Set the isosurface values of the variables "QICE", "QSNOW", "QCLOUD", and "QRAIN" to "0.00005", "0.0005", "0.00005", and "0.0005", respectively. Set all four of the isosurfaces to be colored by W, as we did in figure 5.40.

You can see the isosurfaces of the different variables at the same time in the different visualizers like the following:



<Figure 5.41> Multiple visualizations of 4 different isosurfaces

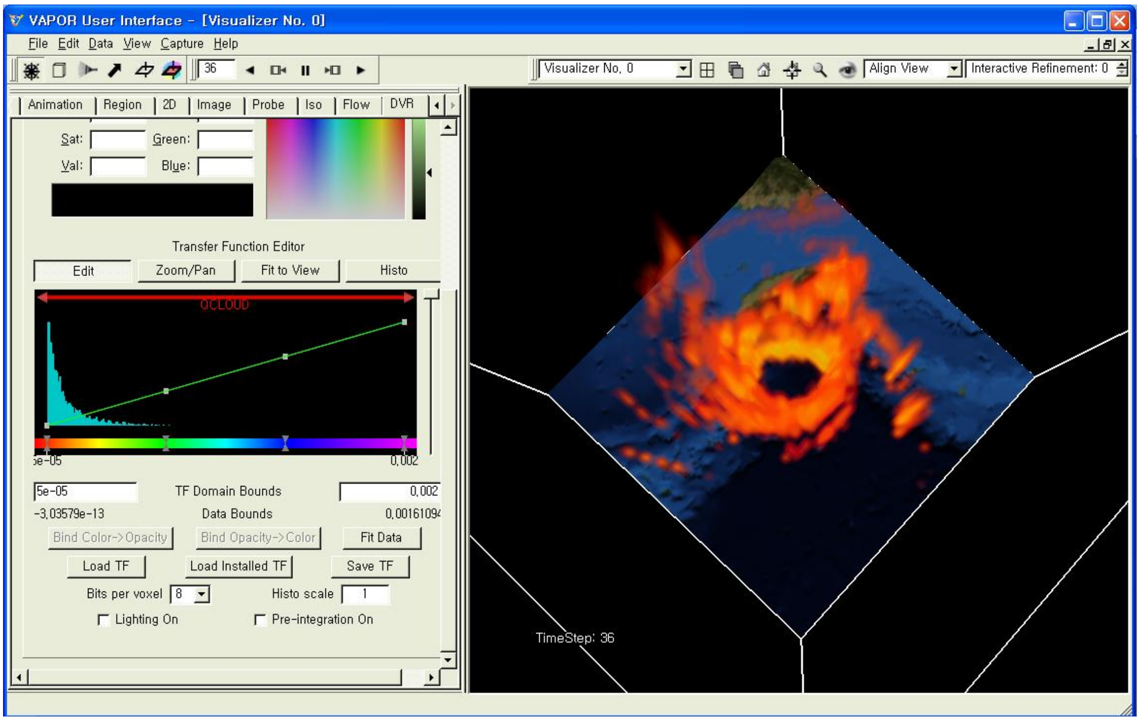
5.6 Volume Visualization

5.6.1 Definition of direct volume rendering

"Direct Volume Rendering" is a technique for directly displaying a sampled 3D scalar field without first fitting geometric primitives to the samples.

In the WRF model output dataset, 3D variables such as "QCLOUD", "QICE", "QSNOW", and "QRAIN" can be easily understood using volume visualization. For example, clouds accompanied by a typhoon can be seen in a volume visualization of "QCLOUD".

Click the "**DVR**" tab. Select the "QCLOUD" variable. Scroll down to the "Transfer Function Editor" window. Recall that we found the isosurface QCLOUD=0.00005 to be useful in visualizing the typhoon cloud. Set "TF Bounds" values to 0.00005 and 0.02. Click the buttons "Fit to View" and "Histo" in the transfer function editor. Enable the DVR renderer. You should see the following:

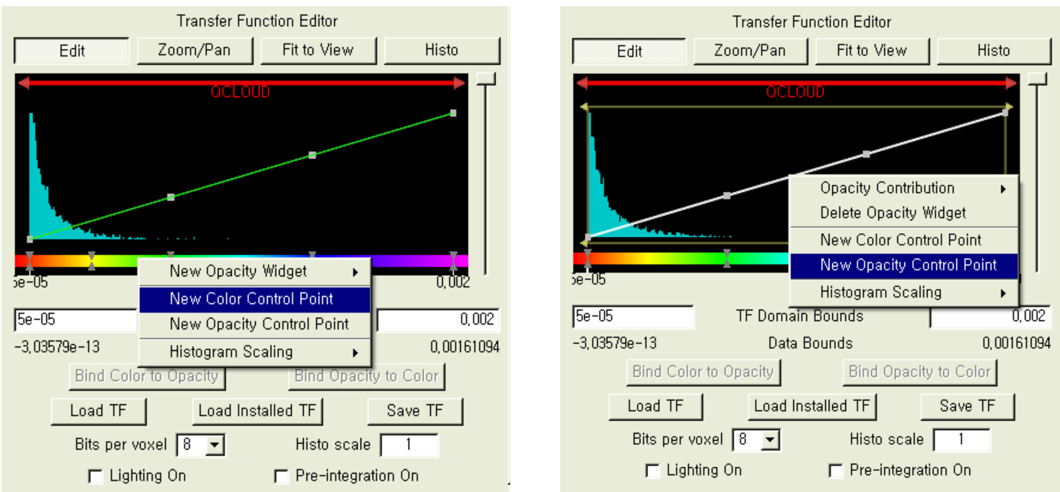


<Figure 5.42> Volume visualization of QCLOUD, time step 36

5.6.2 Use of the transfer function editor

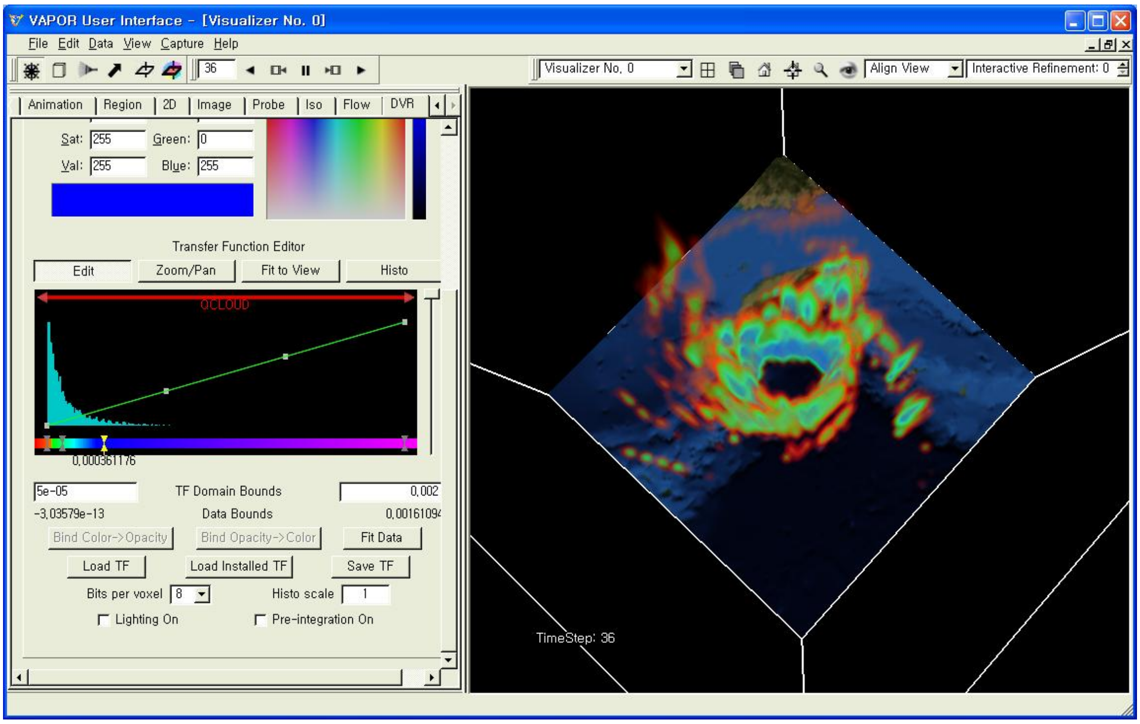
A direct volume renderer requires every sample value to be mapped to opacity and a color. This is done with a "transfer function" which can be a simple ramp, a piecewise linear function or an arbitrary table. Once converted to an RGBA (for red, green, blue, alpha) value, the composed RGBA result is projected on correspondent pixel of the frame buffer. The way this is done depends on the rendering technique.

The appearance (i.e., color and transparency) of the volume rendering is controlled by the "Transfer Function Editor" in the middle of the "DVR" renderer panel. In the center of the "Transfer Function Editor" window, there is a blue-green histogram, indicating the values of the selected variable "QCLOUD". The color is controlled by a horizontal bar of colors at the bottom of the editor. Opacity is controlled by the diagonal green line in the center region. You initially will see four color control points (grey triangle pairs) on the color bar, and four opacity control points (grey squares) on the diagonal green line. Colors and opacity in the scene are interpolated between the control points. You can create and set new control points by clicking in the editor window with the right mouse button, as shown in <Figure 5.43>. You can move these control points by dragging them with the left mouse button or clicking on them with the right mouse button. You can control the overall opacity with the vertical slider on the right side of the transfer function editor. See <Figure 5.44> and <Figure 5.45>.

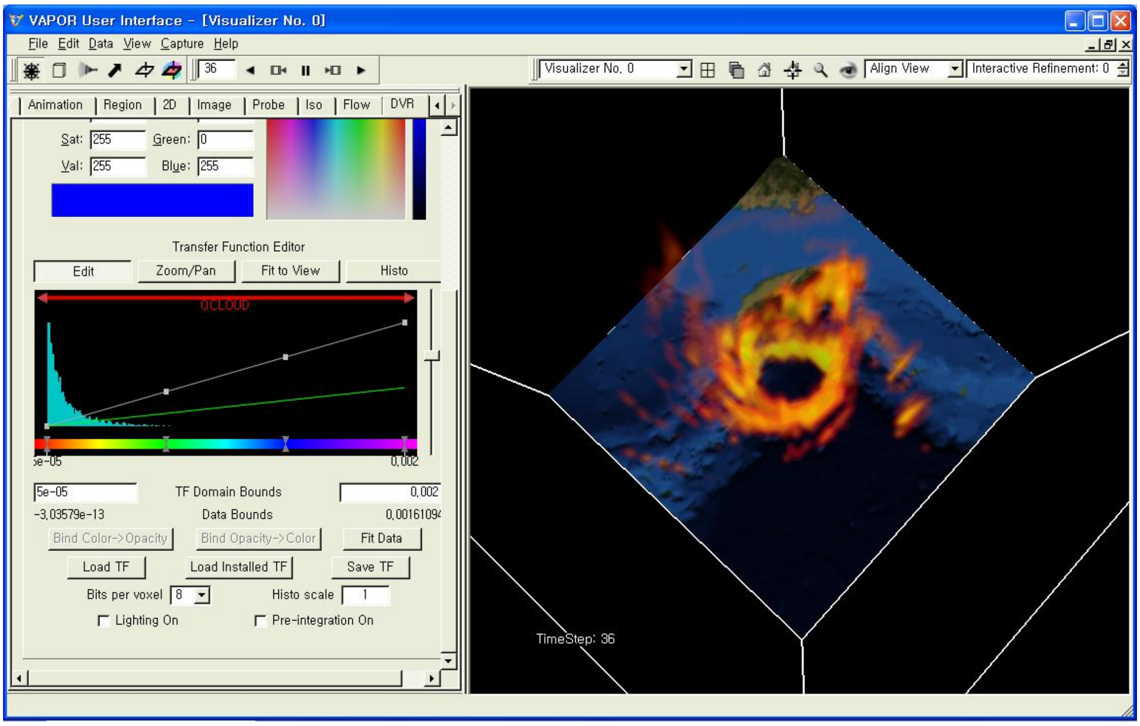


(a) create new color control point (b) create new opacity control point

<Figure 5.43> Creating new color/opacity control points



<Figure 5.44> DVR of QCLOUD with color modified from <Figure 5.42>



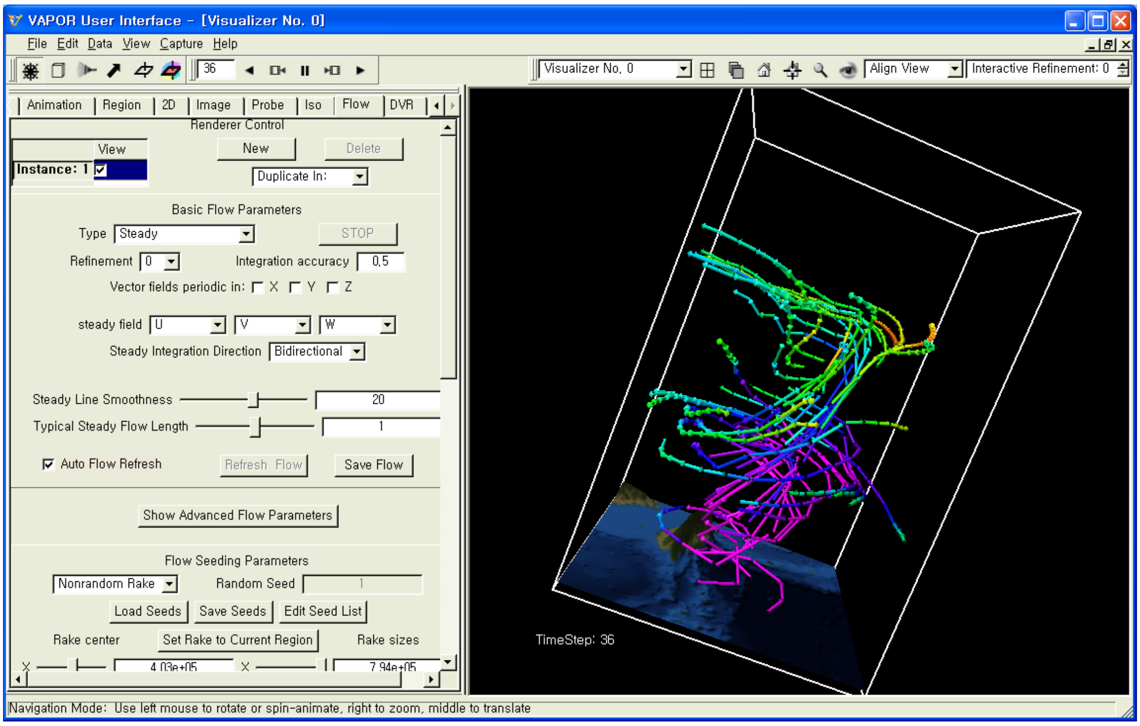
<Figure 5.45> DVR of QCLOUD with opacity modified from <Figure 5.42>

5.7 Probe Visualization

5.7.1 Looking closely at wind with the probe

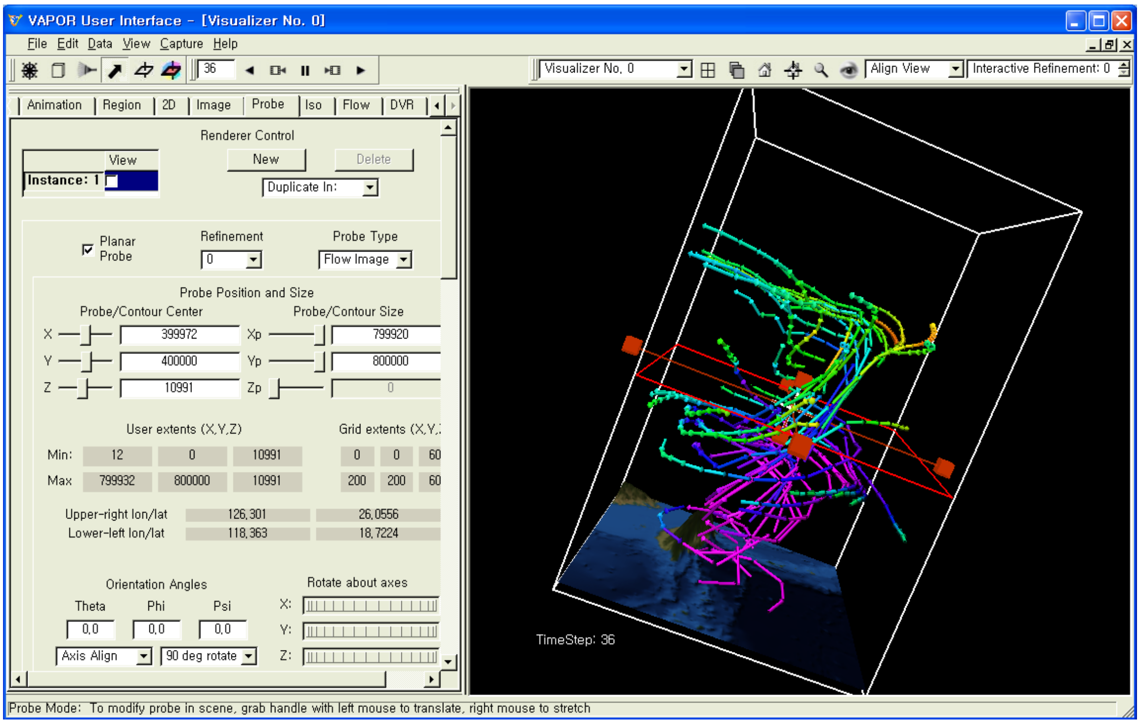
Before inserting the probe into the flow visualization, you'd better reset the visualizer features. Click on "Edit" --> "Edit Visualizer Features". Set the "Scene Stretch Factors X, Y, Z" to "1, 1, 50" to set the vertical stretch factor. Click "OK". Click the "Animation" tab. Set the "Current Frame Position" to 36.

Return to the "**Flow**" renderer panel. Set flow "Type" to "Steady". Scroll down to the "Flow Seeding Parameters". Select "Nonrandom Rake". At the bottom of the "Flow Seeding Parameters" section, type in values 4, 4, 2 into the three text-boxes next to "By dimension X:... Y:... Z:...". This will result in a regularly spaced array of seed points, of sizes 4x4x2 (in the x, y, and z-directions). Your flow settings should be the same as were used in <Figure 5.33> except that you should make the Diameter (in the Shape Parameters) smaller, say 0.3. Make sure your rake is set to the full region. After you enable the flow, you should see the steady flow visualization like the following:



<Figure 5.46> Steady flow visualization at the time step=36

Click the "**Probe**" arrow near the upper-left corner of the VAPOR GUI Window. In the “**Probe**” tab, click the button “Fit to Region”. The probe manipulator will appear in the visualizer window like the following.

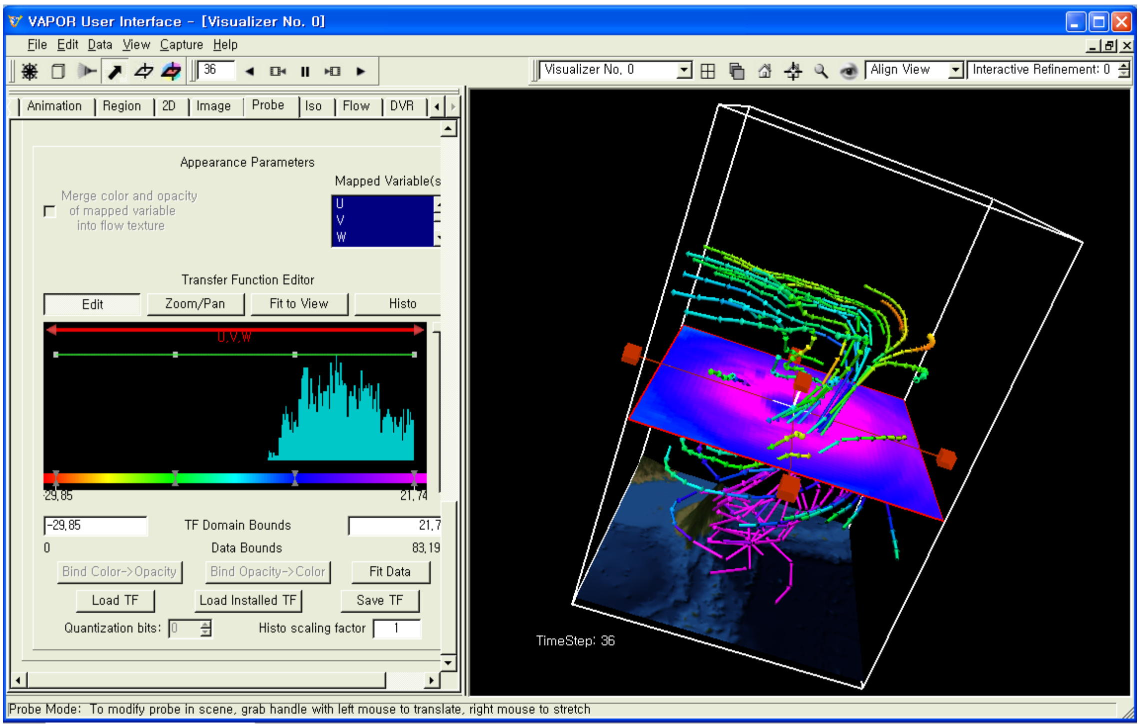


<Figure 5.47> Probe manipulator in the visualizer

By default, the probe is oriented horizontally in the middle of the region. You can move and resize the probe by dragging the manipulator handles with the left (to move) or right (to resize) mouse buttons. When you click with either mouse button on a handle of the probe manipulator, the handle will change color from red to yellow.

At the top of the "**Probe**" panel, select "Data Value" in the "Probe Type" selector. Scroll down to the "Appearance Parameters" section. In the "Mapped Variables" selection box, hold the "Ctrl" key while you select the three variables U, V, and W. (Multiple variable selection results in the norm of the vector of the variables being color-mapped in the probe). This will enable us to see the magnitude of the wind velocity in the cross-section image in the probe.

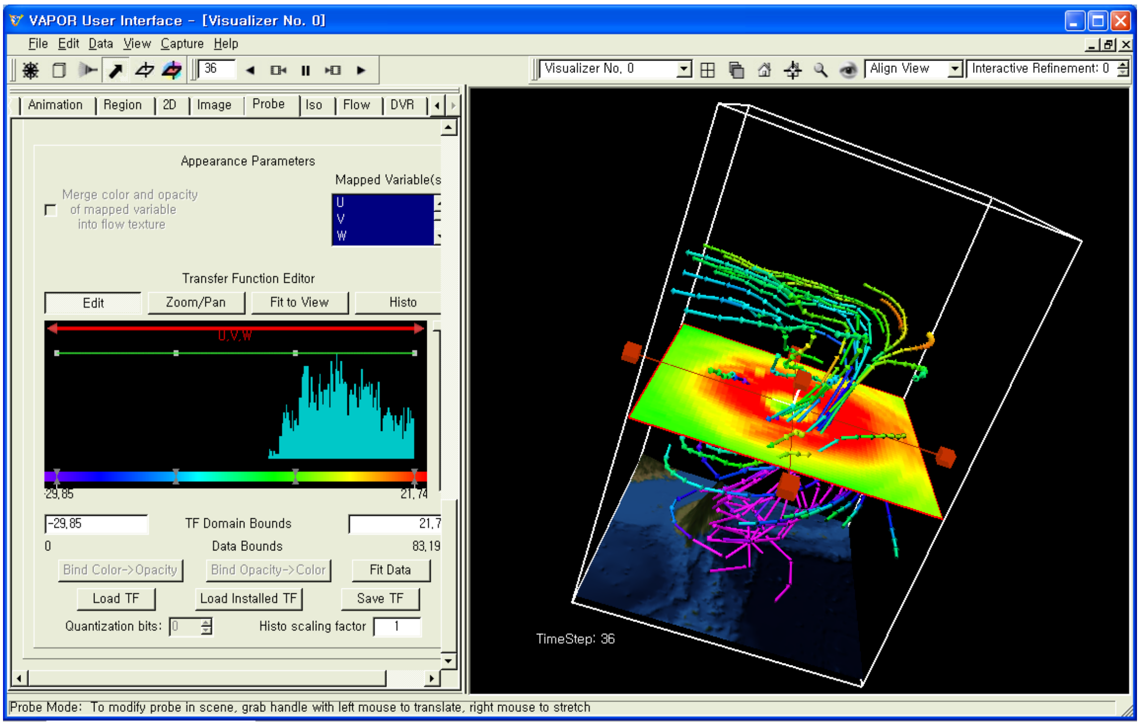
Now check the "Instance: 1" check-box to enable the probe. You will see a horizontal rectangle, colored according to the wind velocity like the following:



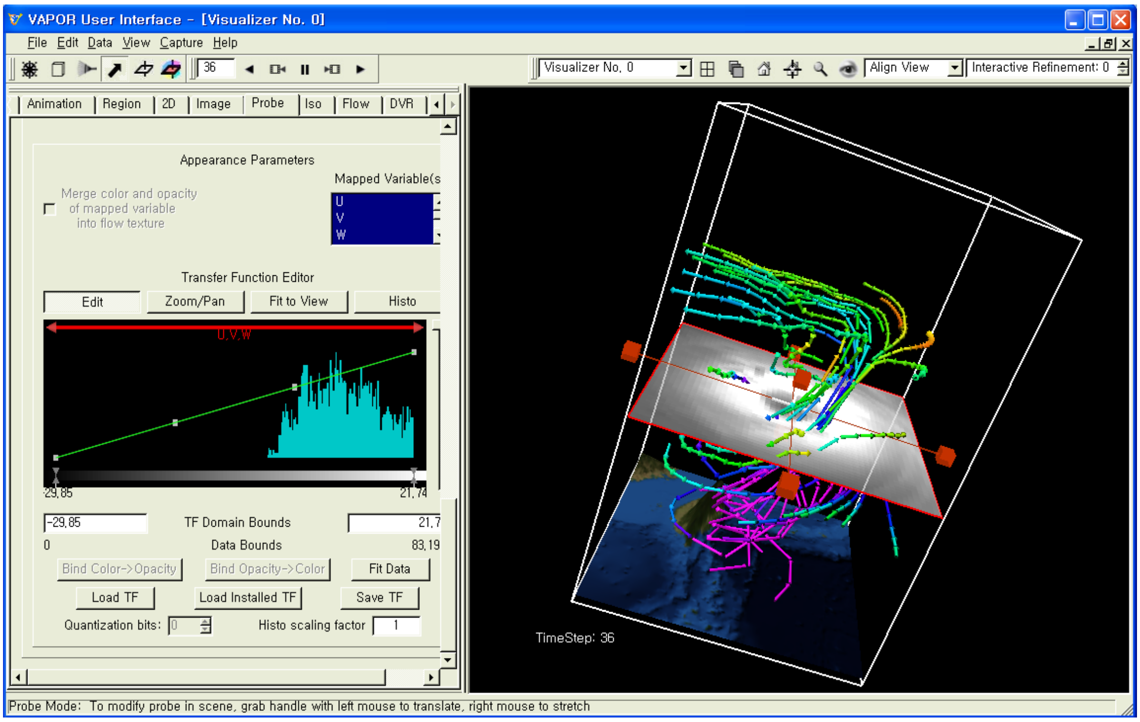
<Figure 5.48> Streamlines as in <Figure 5.46> with a planar probe

By default, large values are colored by violet. In order to map large values of velocity to red, click the "Load Installed TF" and select the transfer function file "reversedOpaque.vtf" in the "Choose the transfer function file to open" dialog that pops up. Then click "Fit to View" and "Histo" above the "Transfer Function Editor" window to see a histogram of the values of the wind velocity in the probe. The image in the probe control box will become a reversed color representation of data values as in the following <Figure 5.49>.

If you select the transfer function file "grayscale.vtf" instead of "reversedOpaque.vtf", the image in the probe will become a partially transparent black-and-white representation of data values as in <Figure 5.50>.

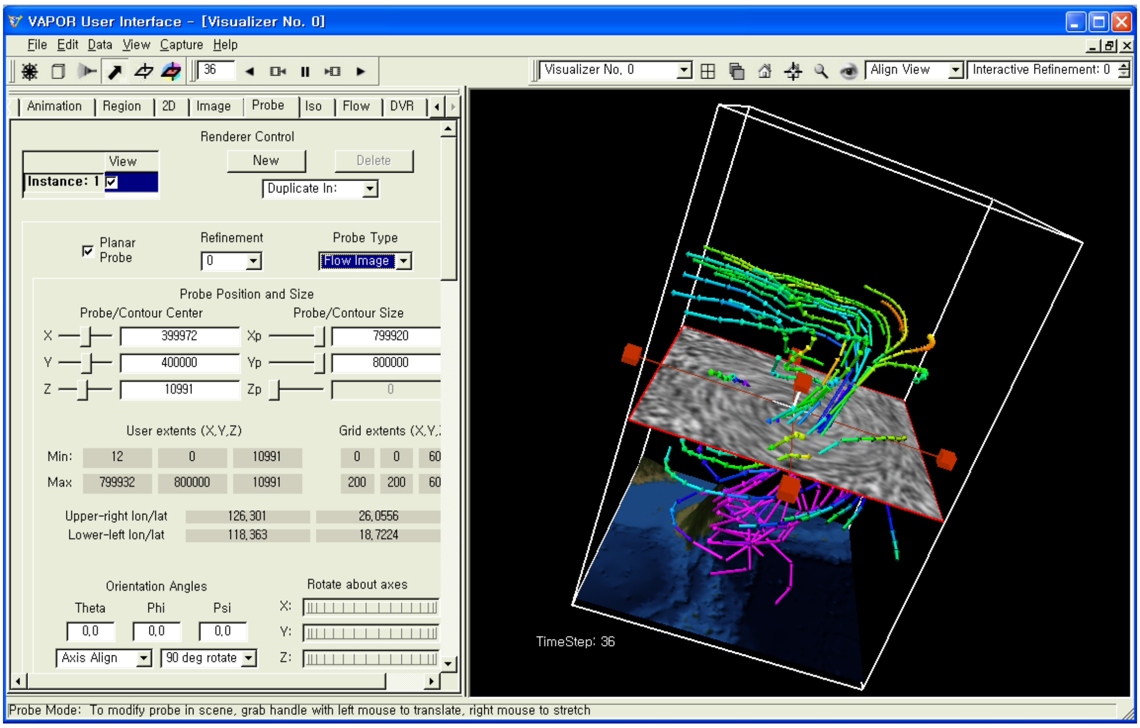


<Figure 5.49> Probe colored by a color-reversed mapping



<Figure 5.50> Probe colored by black-and-white mapping

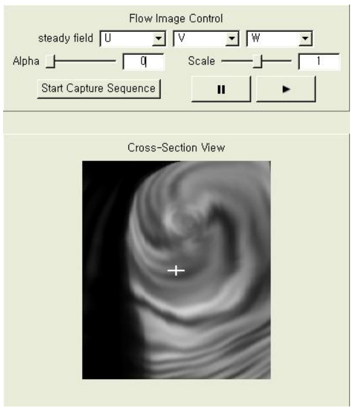
You can change the image in the probe control box. At the top of the "Probe" panel, select "Flow Image" in the "Probe Type" selector. The image in the probe control box will become a black-and-white representation of flowing particles as in the following:



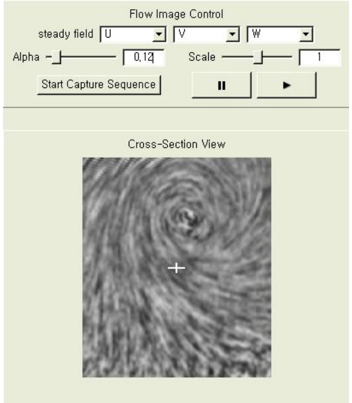
<Figure 5.51> Probe with Type="flow image"

The flow image in the probe is best understood by animating it. In the "Flow Image Control" section of the "**Probe**" panel, above the "Cross Section View", click on the play button. You can see the flow motion represented as moving particles.

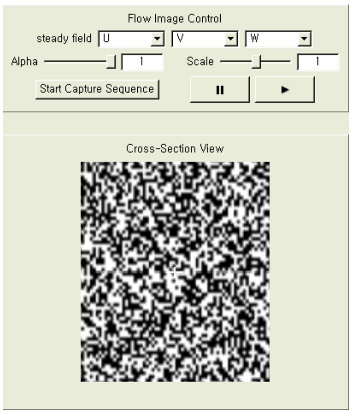
VAPOR uses a flow visualization technique known as "Image Based Flow Visualization" or IBFV, invented by Jarke Van Wijk (Proc. Siggraph 2002). Basically it works by advecting random noise in the velocity field. The flow motion is projected into the plane of the probe. You can tune the flow images by tweaking the "Scale" and "Alpha" sliders in the "Flow Image Control", and the flow images can be captured as a sequence of JPEG files by pressing the "Start Image Capture" button.



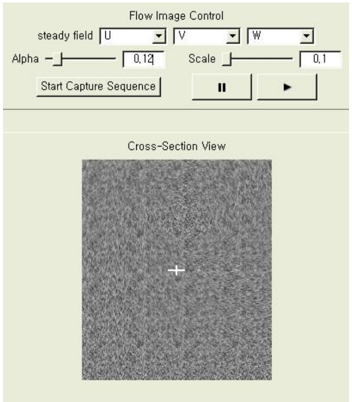
(a) Alpha=0 (min)



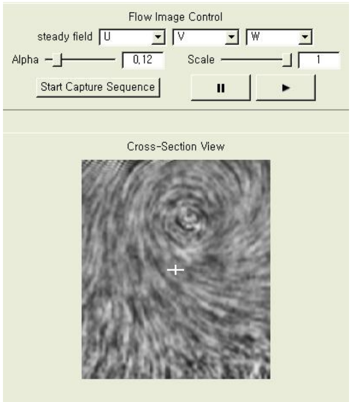
(b) Alpha=0.12 (default)



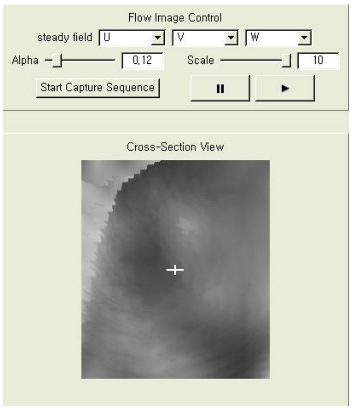
(c) Alpha=1 (max)



(d) Scale=0.1 (min)



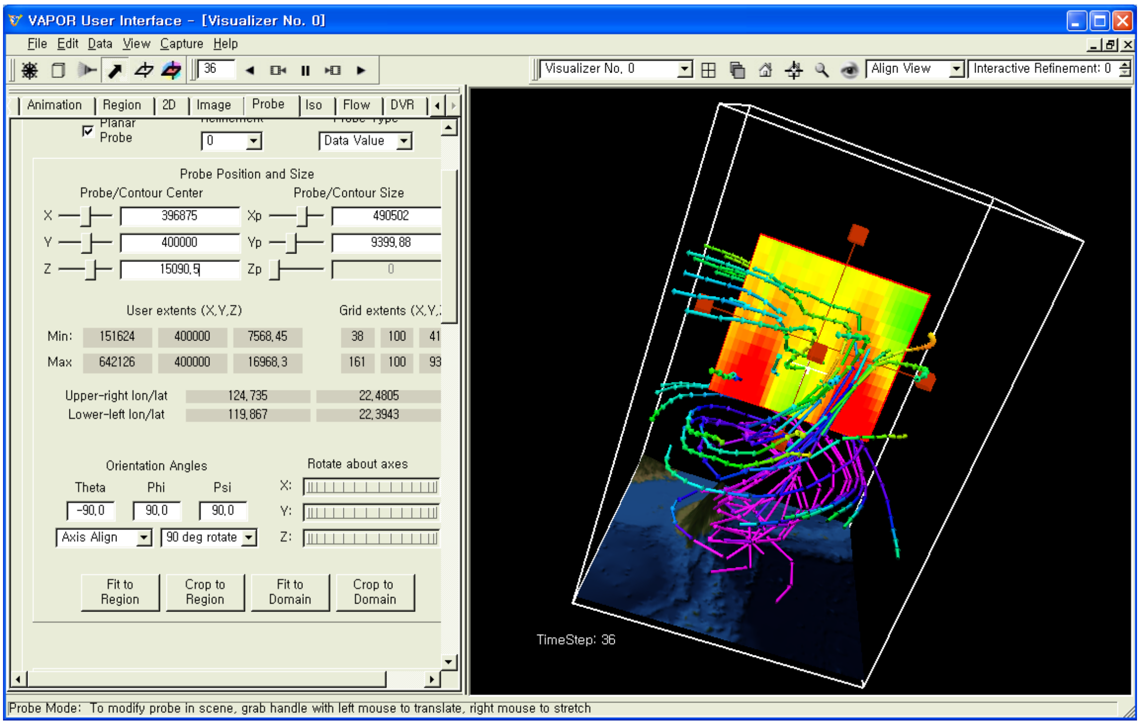
(e) Scale=1 (default)



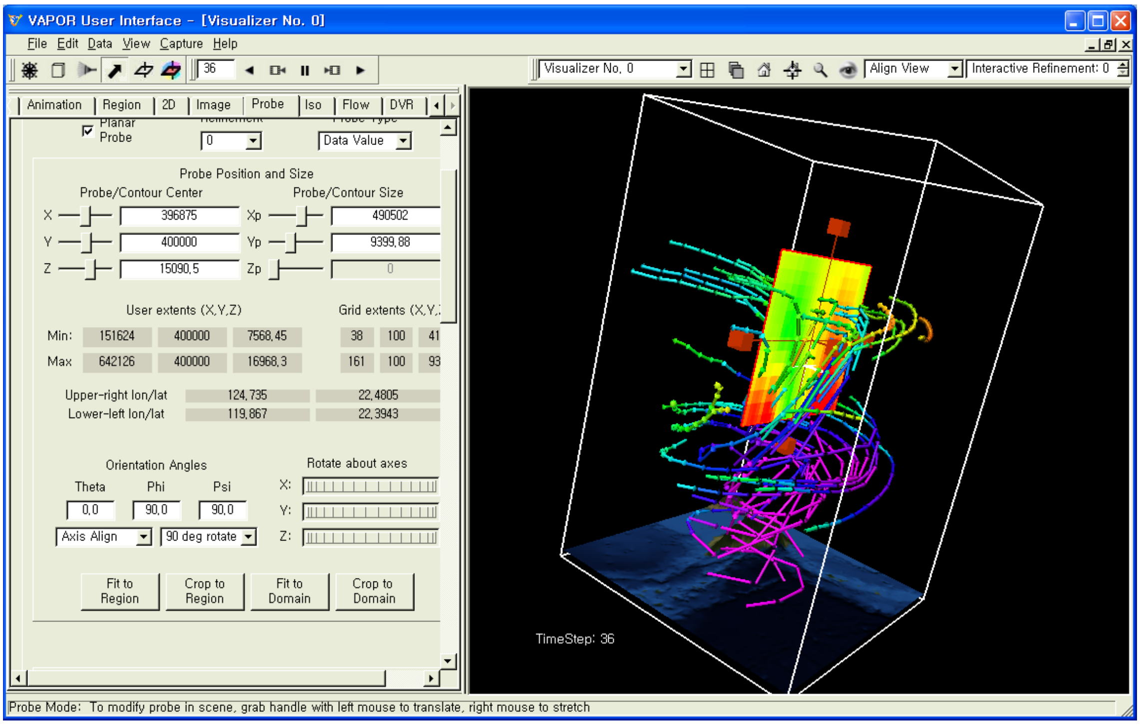
(f) Scale=10 (max)

<Figure 5.52> Comparison of cross-section view in the flow images

We can orient the probe vertically to see the vertical wind motion. Reset the “Probe Type” selector to “Data Value” and reload the installed “reversedOpaque.vtf” transfer function. In the "**Probe**" panel, at the bottom of the section entitled "Probe Position and Size", there is a selector displaying the text "90 degree rotate". From that selector, choose "+X" to rotate the probe 90 degrees about the positive X axis. Then click the button “Fit to Region” below that selector. Shrink the probe size, dragging the probe handles with the right mouse button pressed, making the probe about the same size as is shown in <Figure 5.53>. Using the “90 Degree Rotate” selector, choose "+Z" to rotate the probe 90 degrees about the positive Z axis. The "Cross-Section View" in the "**Probe**" panel is as seen from the east side. To facilitate navigating around the probe, click the button labeled "Copy selected point to: View Center", making the center of your view be the selected point (center) in the probe.



(a) rotate the probe 90 degrees about the positive X axis

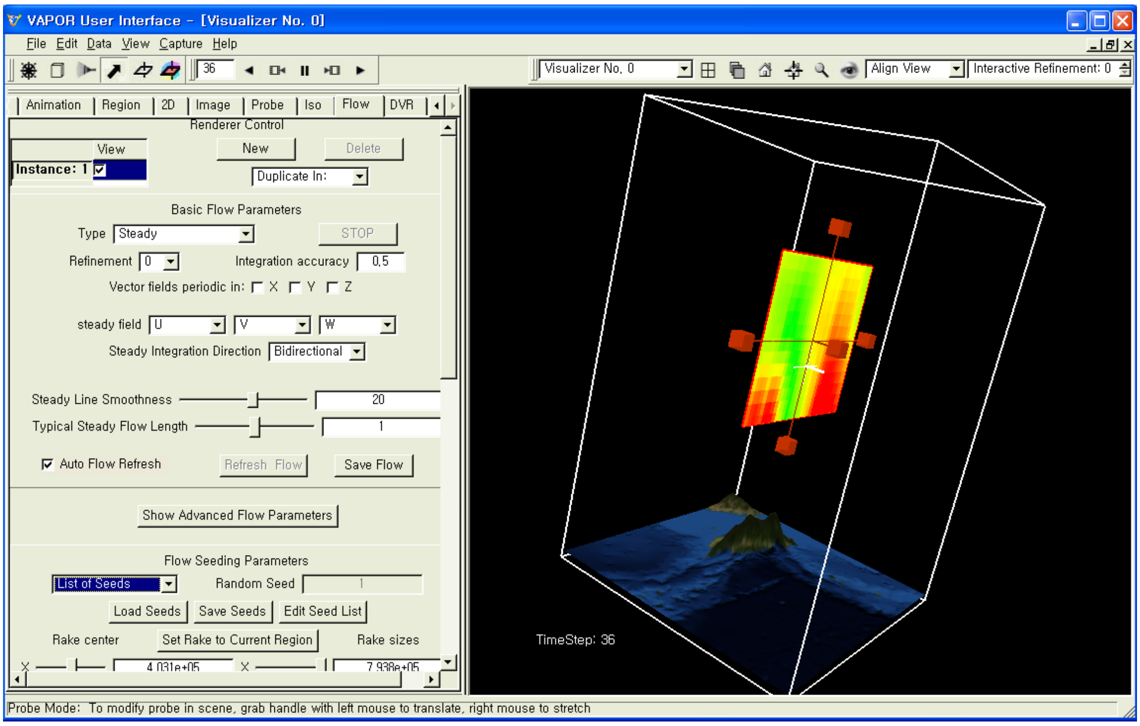


(b) rotate the probe 90 degrees about the positive Z axis

<Figure 5.53> Orienting the probe vertically

5.7.2 Placing flow seeds with the probe

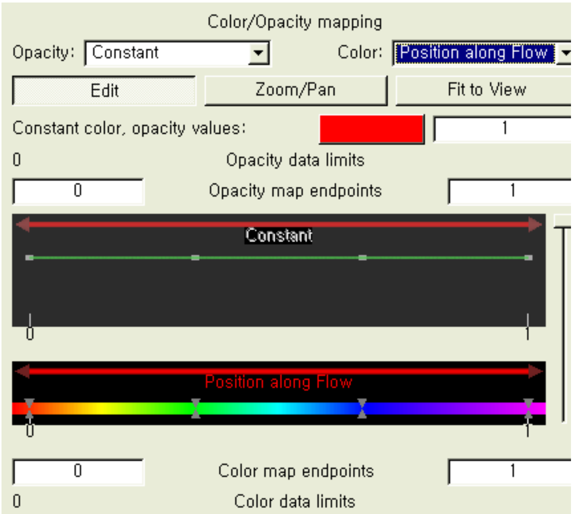
You can place individual seed points manually, using the probe. In the "**Flow**" renderer panel, under "Flow Seeding Parameters" section, select "List of Seeds". You will notice that the streamlines have disappeared because there are no seeds in the current seed list. There will be a couple of error messages indicating that there are no flow seeds. The flow should already be enabled.



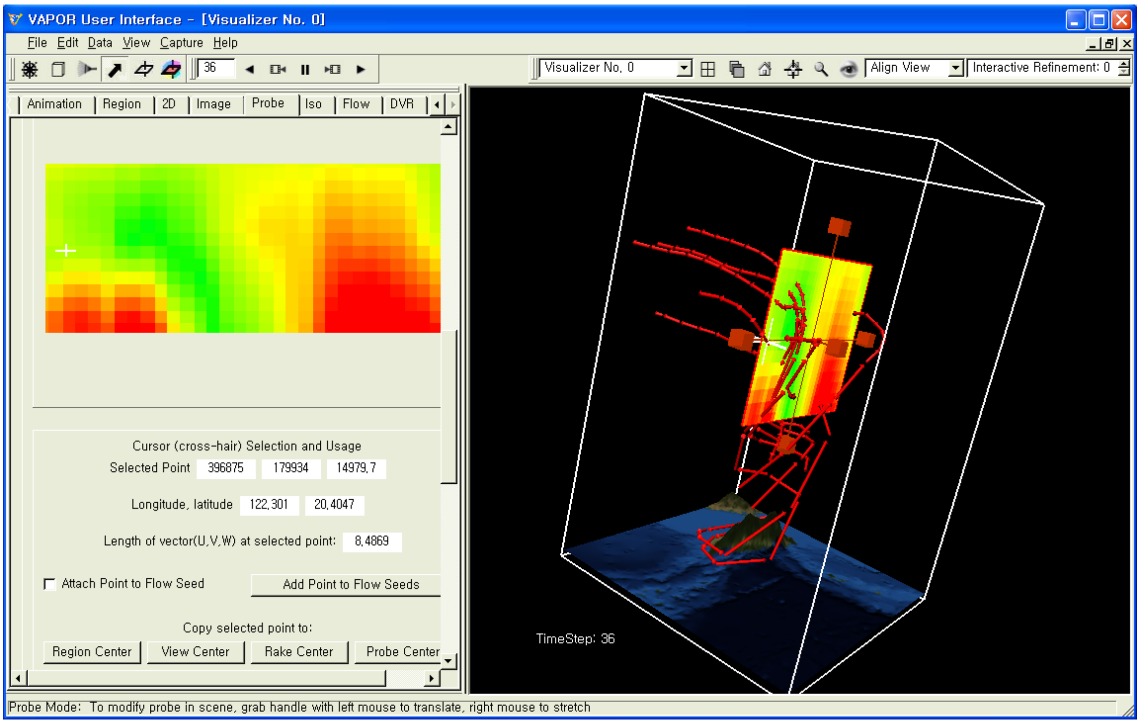
<Figure 5.54> Orienting the probe vertically without flow seeds

Click the "**Probe**" tab and scroll down where you can see the options below the "Cross-Section View". Check the check-box entitled "Attach Point to Flow Seed". You will see a streamline originating at the position of the cross-hairs in the probe. By moving the cursor around the probe image you can position flow seeds and see the resulting streamlines. Each time you find a streamline of interest, click the button "Add Point to Flow Seeds". Repeat this until you have added enough seeds to get an understanding of the wind flow in this region.

At the bottom of the "**Flow**" renderer panel, in the section "Color/ Opacity mapping", select the color mapping to be "Position along Flow" instead of "Field Magnitude", so that streamline color changes along the length of each flow line, going from red to violet as in the following <Figure 5.55>.

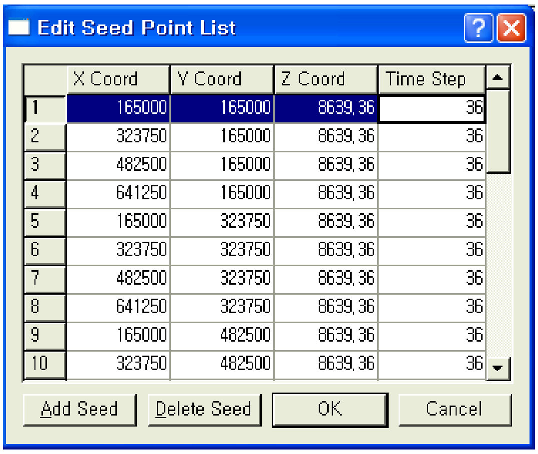


<Figure 5.55> Color/Opacity mapping section



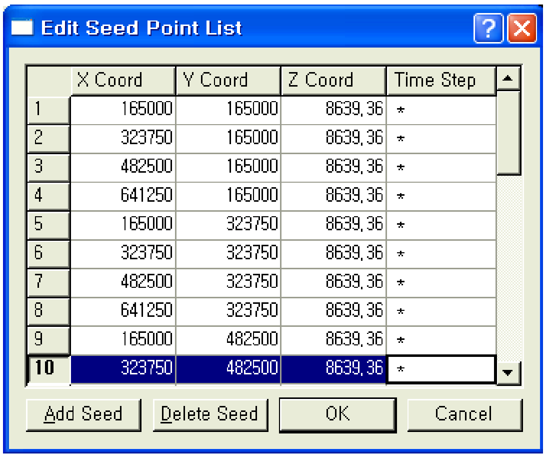
<Figure 5.56> Flow visualization using seeds selected in the probe

You can modify the values of seed points by clicking "Edit Seed List". Note that each seed has four coordinates. If you set the probe to be parallel to the X-Y plane, the third(Z) coordinate has the same values as in <Figure 5.57>. If you set the probe to be parallel to the Y-Z plane, the first(X) coordinate has the same values as in <Figure 5.58>. The fourth coordinate is the value corresponding to the time step we are currently visualizing. To specify seed points that are valid for all time steps, set the time step to "*" or "-1".



	X Coord	Y Coord	Z Coord	Time Step
1	165000	165000	8639.36	36
2	323750	165000	8639.36	36
3	482500	165000	8639.36	36
4	641250	165000	8639.36	36
5	165000	323750	8639.36	36
6	323750	323750	8639.36	36
7	482500	323750	8639.36	36
8	641250	323750	8639.36	36
9	165000	482500	8639.36	36
10	323750	482500	8639.36	36

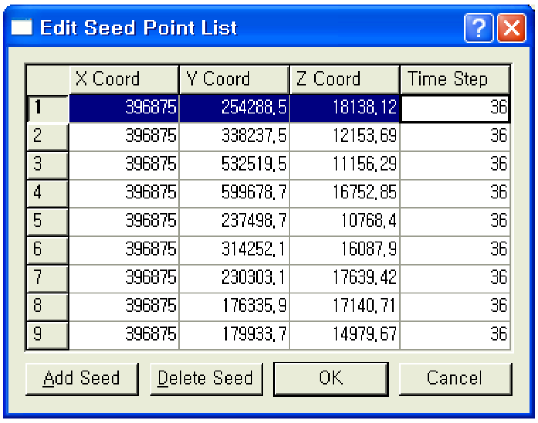
(a) before modifying time step



	X Coord	Y Coord	Z Coord	Time Step
1	165000	165000	8639.36	*
2	323750	165000	8639.36	*
3	482500	165000	8639.36	*
4	641250	165000	8639.36	*
5	165000	323750	8639.36	*
6	323750	323750	8639.36	*
7	482500	323750	8639.36	*
8	641250	323750	8639.36	*
9	165000	482500	8639.36	*
10	323750	482500	8639.36	*

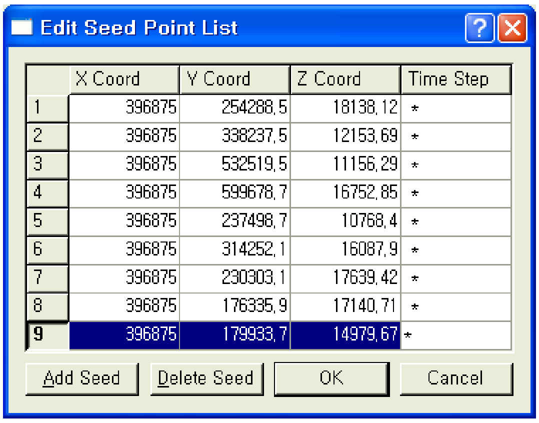
(b) after modifying time step

<Figure 5.57> Seed list for <Figure 5.49>



	X Coord	Y Coord	Z Coord	Time Step
1	396875	254288.5	18138.12	36
2	396875	338237.5	12153.69	36
3	396875	532519.5	11156.29	36
4	396875	599678.7	16752.85	36
5	396875	237498.7	10768.4	36
6	396875	314252.1	16087.9	36
7	396875	230303.1	17639.42	36
8	396875	176335.9	17140.71	36
9	396875	179933.7	14979.67	36

(a) before modifying time step



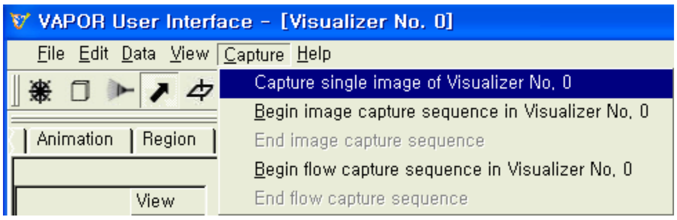
	X Coord	Y Coord	Z Coord	Time Step
1	396875	254288.5	18138.12	*
2	396875	338237.5	12153.69	*
3	396875	532519.5	11156.29	*
4	396875	599678.7	16752.85	*
5	396875	237498.7	10768.4	*
6	396875	314252.1	16087.9	*
7	396875	230303.1	17639.42	*
8	396875	176335.9	17140.71	*
9	396875	179933.7	14979.67	*

(b) after modifying time step

<Figure 5.58> Seed list for <Figure 5.56>

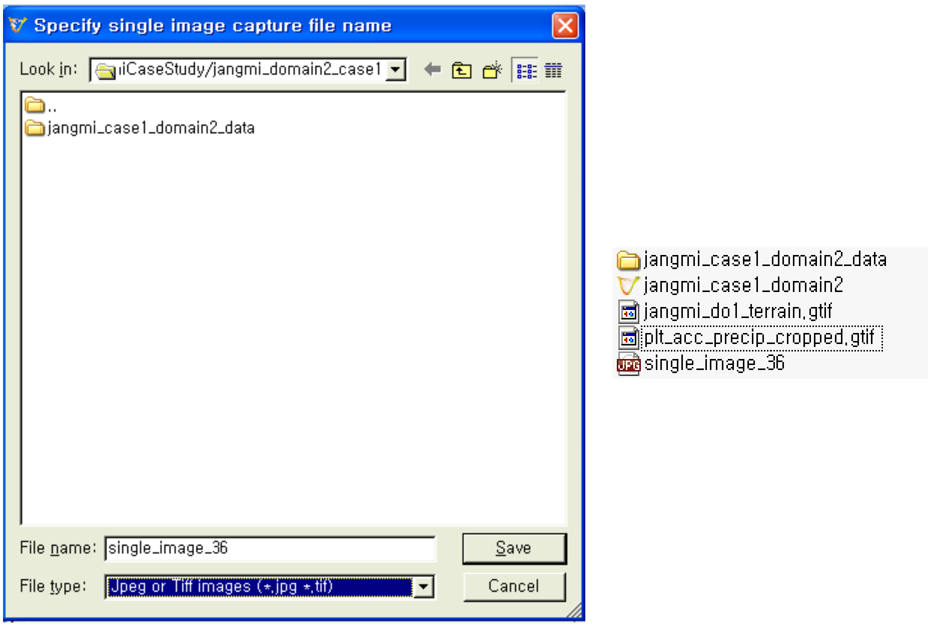
5.8 Capturing a single image file and animated files

VAPOR supports capturing a single image of the visualizer. Set the "Current Frame Position" to 36 in the "**Animation**" panel. (You can also set the time step in the animation toolbar, next to the mouse mode icons.) Click "**Capture**" and then select "Capture single image of Visualizer No. 0".



<Figure 5.59> Capturing single image

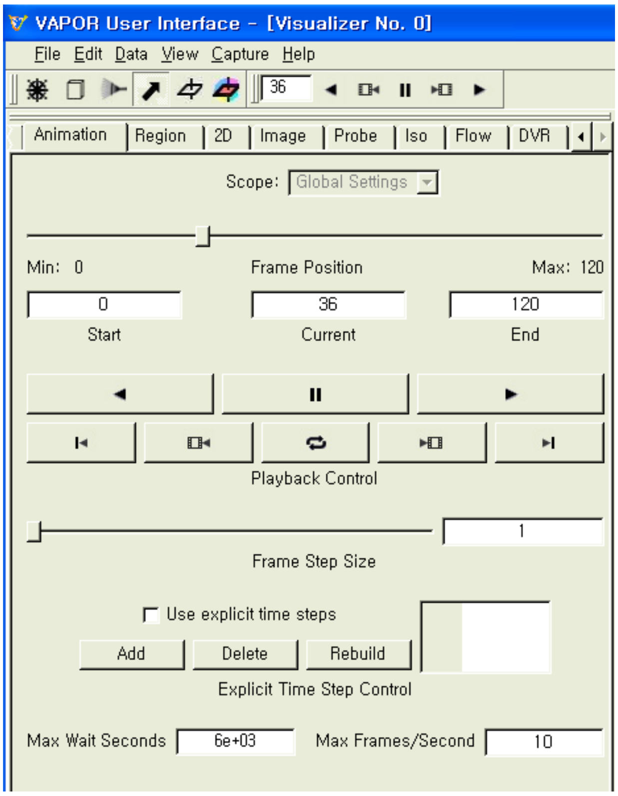
From the "Specify single image capture file name" dialog, select a directory and a file name where the resulting JPEG file or tiff file will be saved. If you specify a filename that ends with ".tif" it will save a tiff file; otherwise the file saved will be a JPEG. (Note: tiff images are useful if you want an uncompressed image.)



(a) specify a file name (b) resulting JPEG or tiff file

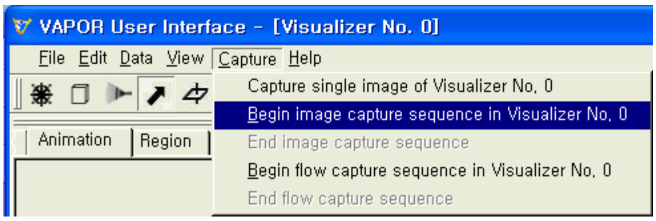
<Figure 5.60> Specify a file name for a single image

VAPOR supports capturing animation sequences. Set the "Start" frame position to 36, "Current" frame position to 36, and "End" frame position" to 41 in the "Animation" panel.



<Figure 5.61> Animation panel

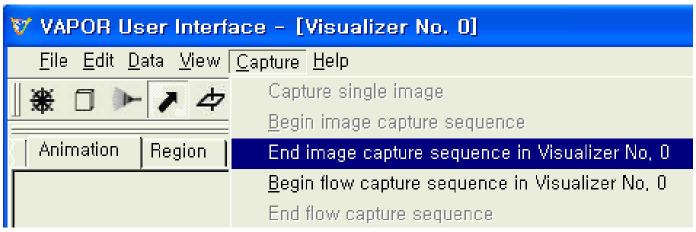
Click "**Capture**" and then select "Begin image capture sequence in Visualizer 0".



<Figure 5.62> Beginning image capture sequence

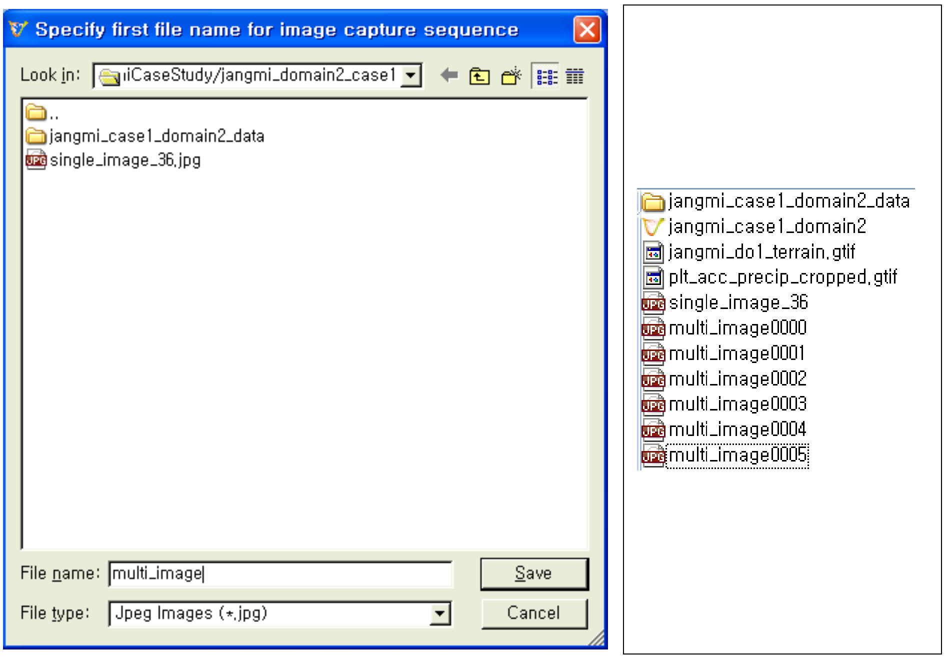
From the file selection dialog, select a directory and a file name where the resulting JPEG files will be saved. Then, in the "**Animation**" panel, play forward to the end of the sequence (click play button).

After the sequence is complete, click "End image capture sequence in Visualizer 0" from the Capture menu.



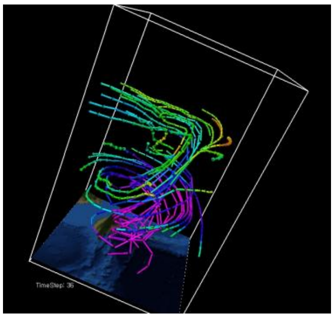
<Figure 5.63> Ending image capture sequence

You will see that a series of JPEG files has been saved. These may be converted to QuickTime or Window Media Player files using software that you can purchase or obtain from the Web.

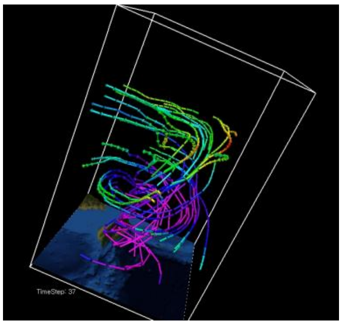


(a) specify a file name (b) resulting JPEG files

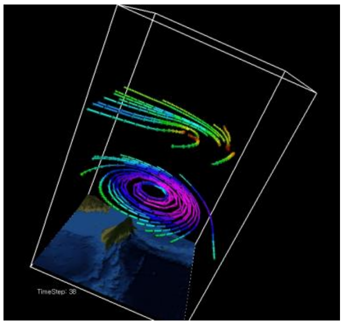
<Figure 5.64> Specify a file name for multiple images



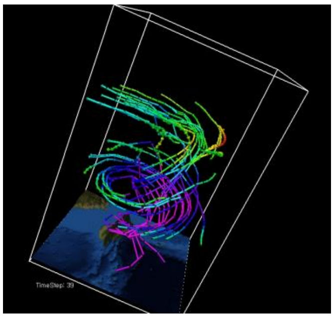
(a) time step=36



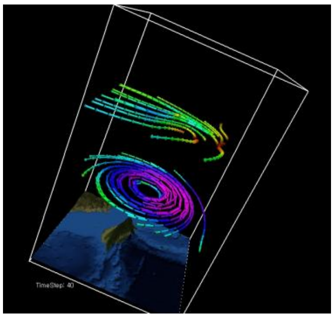
(b) time step=37



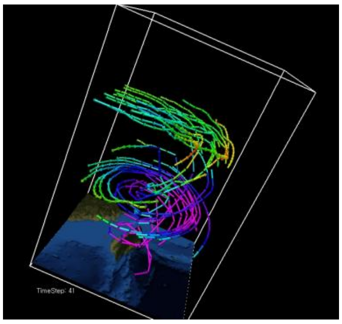
(c) time step=38



(d) time step=39



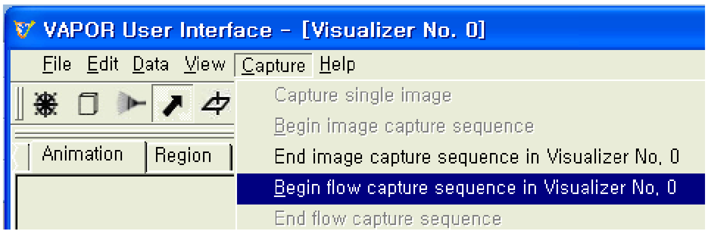
(e) time step=40



(e) time step=41

<Figure 5.65> Sample images saved as JPEG files

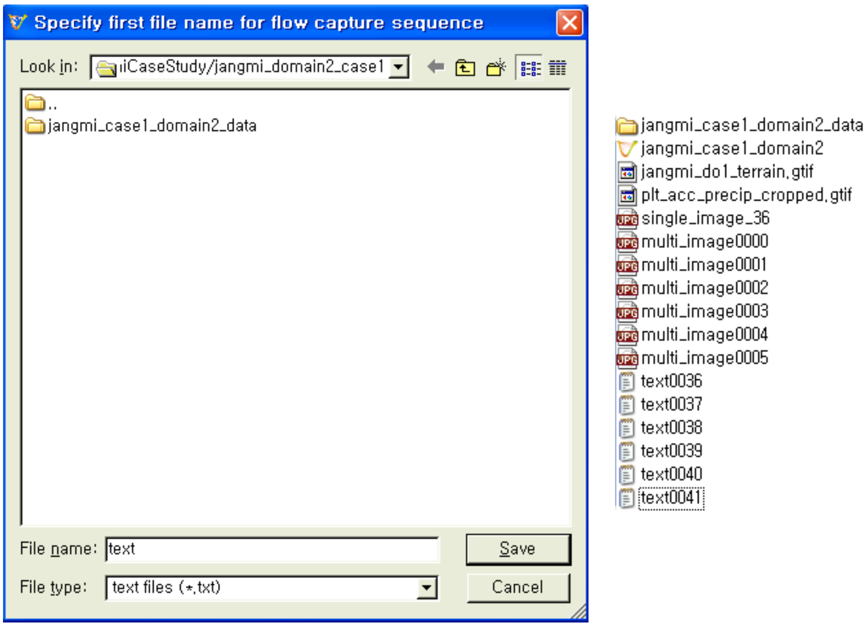
VAPOR supports saving the points along steady flow lines (streamlines) during an animation sequence. Click "**Capture**" and then select "Begin flow capture sequence in Visualizer No. 0".



<Figure 5.66> Beginning flow capture sequence

From the file selection dialog, select a directory and a file name where the resulting TXT files will be saved. Then, in the "**Animation**" panel, play forward to the end of the sequence (click play button).

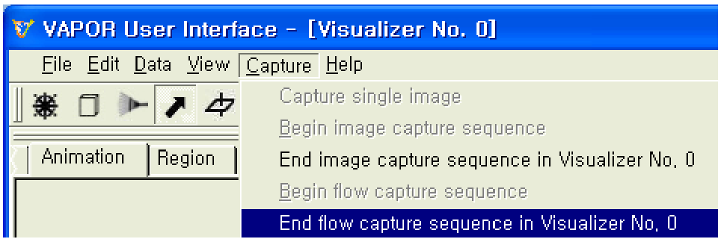
You will see that a series of TXT files has been saved. These files can be read with a text editor such as Wordpad.



(a) specify a file name (b) resulting TXT files

<Figure 5.67> Specify a file name for flow

After the sequence is complete, click the "End capture sequence in Visualizer 0".



<Figure 5.68> Ending flow capture sequence

In the example below, the first line means that time step is 41, the number of points in the following flow line (streamline) is 19, and the seed point of that streamline is at position 9 in the sequence. Following the first line are 19 lines indicating the sequence of (19) points in the streamline. Each point in the streamline has three coordinates. The first column is X-coordinate, the second column is Y-coordinate, and the third column is Z-coordinate.

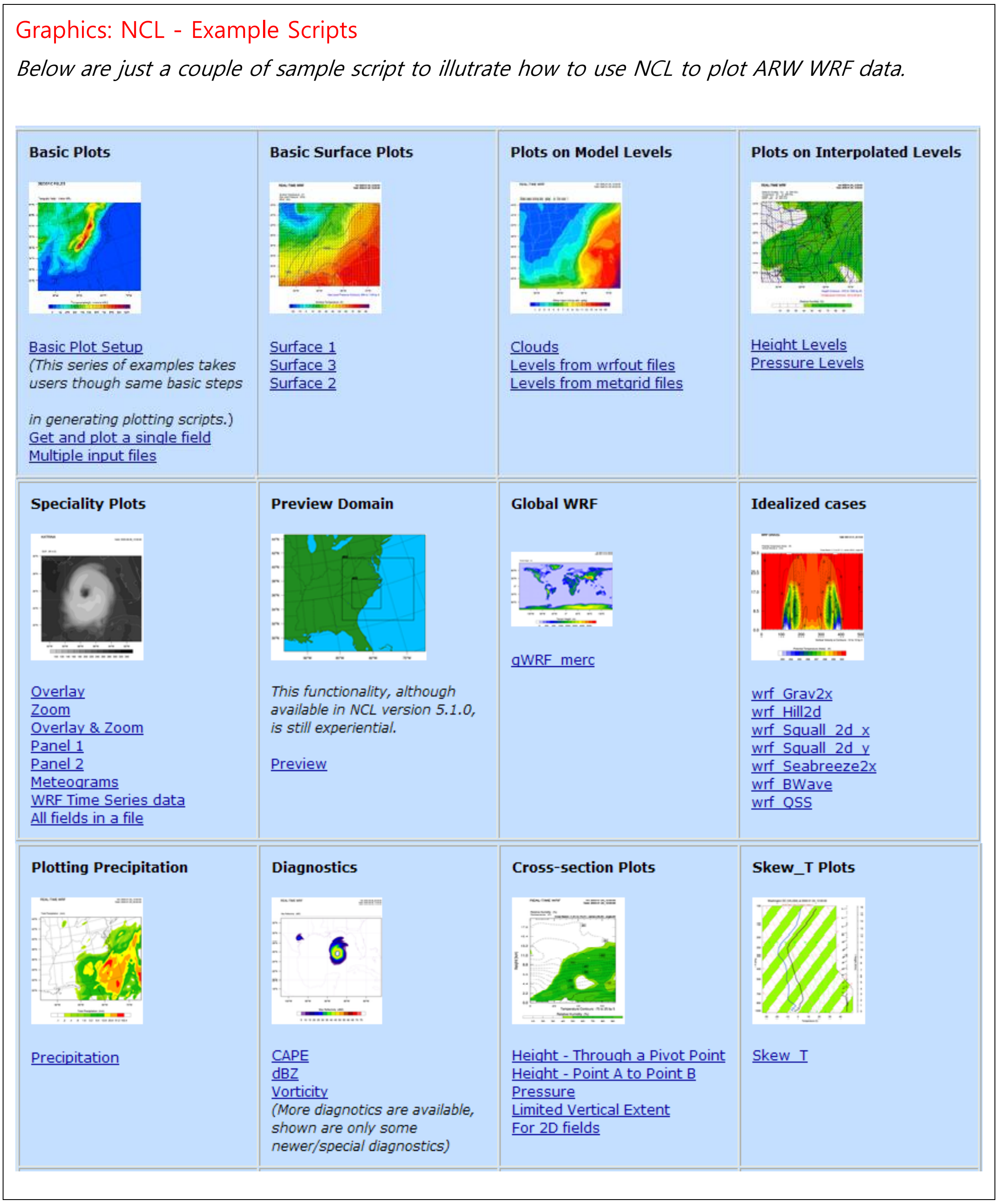
```
41 19 9
    378449     611071     431467
    285898     585795     453379
    218562     540225     454992
    174823     481974     461919
    149072     420364     443545
    135129     357940     440378
    131870     297703     439482
    137135     244918     439028
    147931     200985     436398
    164800     164800     432786
    187274     134774     432880
    211143     108132     437357
    237230    85819.1     452657
    265236    68046.8     475225
    293058    53347.8     497817
    319875    39508.1     513515
    346288    26765.7     517651
    373261    17017.9     524056
    401482    12177.1     541010
41 19 9
    552459     274863     293848
    575958     408808     314236
    497117     539528     364212
    371139     569027     366431
    273961     517580     400828
    224622     434165     400392
```

<Figure 5.69> Contents of a text file saved by flow capture

[Appendix] Making georeferenced 2D images

1. Identify the NCL plot to insert into the VAPOR scene

(1) For examples of such NCL plots, check the example scripts at the "WRF_ARW Online Tutorial" Website.
http://www.mmm.ucar.edu/wrf/OnLineTutorial/Graphics/NCL/NCL_examples.htm



(2) Select the example script to fit your purpose.

A typical NCL script for plotting WRF data is of the form:

```
; load supporting libraries...
load ".........."

; open/load files, etc...
..........

; open workstation...
wks=gsn_open_wks("ps", "wrfPlot")

; set up resources, generate contours, etc...
..........

; generate a plot...
plot=wrf_map_overlays(wrfFile, wks, (/countor/), pltres, mpres)


...........

; clean up...
```

It is also common to see the plot-generation segment of code inside of a loop over time or over a collection of files:

```
; open workstations...
wks=gsn_open_wks("ps", "wrfPlot")
..........

do it=0, numTimes-1


..........
plot=wrf_map_overlays(wrfFile, 잔, (/contour/), pltres, mpres)
..........
end do


..........
```

2. Modify the selected NCL script to produce geoTiff output

There are four minor modifications that must be made to an existing WRF-NCL script in order to capture geoTiff output, as indicated below:

```
; load supporting libraries...
load "...path_to_supporting_scripts.../wrf2geotiff.ncl"

; open workstation...
wks=gsn_open_wks("ps", "wrfPlot")

; initialize the geotiff-capture process...
wrf2gtiff=wrf2geotiff_open(wks)
..........

times = wrf_user_list_times(wrfFile)

do it=0,numTimes-1
..........
plot=wrf_map_overlays(wrfFile, wks, (/contour/), pltres, mpres)
; write the plot to the geotiff file (crop to bounds)...
wrf2geotiff_write(wrf2gtiff, wrfFile, times(it), wks, plot, True)
..........
end do
..........
; close the geotiff file...
wrf2geotiff_close(wrf2gtiff, wks)
```

The below is an example script to plot accumulated precipitation.

```
load "$NCARG_ROOT/lib/ncarg/nclscripts/csm/gsn_code.ncl"
load "$NCARG_ROOT/lib/ncarg/nclscripts/wrf/WRFUserARW.ncl"
load "$NCARG_ROOT/lib/ncarg/nclscripts/csm/shea_util.ncl"
load "$VAPOR_HOME/share/vapor-1.4.2/examples/NCL/wrf2geotiff.ncl"
```

```
begin

; Make a list of all files we are interested in
DATADir = "/gpfs/proj2/DASG/alan/minsu/"
wrfFILES = systemfunc (" ls -1 " + DATADir + "wrfout_d02_2008-09-2[678]*")
numFiles = dimsizes(wrfFILES)

; We generate plots, but what kind do we prefer?
; type = "x11"
; type = "pdf"
type = "ps"

wks = gsn_open_wks(type,"plt_acc_precip")

gsn_define_colormap(wks,"BlAqGrYeOrReVi200") ; overwrite the .hluresfile color map

; initialize our tiff-capture process...
wrf2gtiff = wrf2geotiff_open(wks)

; Set some basic resources
res = True
res@MainTitle = "REAL-TIME WRF"

pltres = True
mpres = True
mpres@gsnPaperOrientation = "portrait"

; linestyle resources...
lnres = True
lnres@gsLineThicknessF = 3.0

; text-annotation resources
txres = True
txres@txFont = "helvetica-bold"
txres@txFontColor = "Black"
txres@txFontHeightF = 0.01
txres@txJust = "TopLeft"
```

```
; plot resources...
pltres@FramePlot = False ; we want manual control over plot frame...
pltres@gsnPaperOrientation = "portrait"
pltres@gsnMaximize = False

do i=0,numFiles-1
wrfFILES(i) = wrfFILES(i) + ".nc"
end do
inpFiles = addfiles(wrfFILES,"r")

; Loop over our input files, generating a plot for each timestamp
do i=0,numFiles-1
wrfFile = inpFiles[i]

; What times and how many time steps are in the data set?
times = wrf_user_list_times(wrfFile) ; get times in the file
ntimes = dimsizes(times) ; number of times in the file

; Time loop...
do it = 0,ntimes-1
print("Working on time: " + times(it) )
res@TimeLabel = times(it) ; Set Valid time to use on plots

; First get the variables we will need
rainnc = wrf_user_getvar(wrfFile,"RAINNC",it)

; Plotting options for RAINNC
opts = res
opts@cnFillOn = True
opts@gsnSpreadColorEnd = -27
contour = wrf_contour(wrfFile, wks, rainnc, opts)

; MAKE PLOT
pltres@FramePlot = False ; do not frame plot - will do this manually later
pltres@gsnPaperOrientation = "portrait"
pltres@gsnMaximize = False
plot = wrf_map_overlays(wrfFile,wks,(/contour/),pltres, mpres)
```

```
; for cropped image, set True ; for uncropped image, set False)
wrf2geotiff_write(wrf2gtiff, wrfFile, times(it), wks, plot, True)
frame(wks) ; Now that we are done drawing, draw the frame

end do ; END OF TIME LOOP
end do ; END OF FILE LOOP

wrf2geotiff_close(wrf2gtiff, wks)

end
```

3. Run NCL scripts

(1) Download NCL binaries and/or source code

NCL is available for free as pre-compiled binaries for various systems or as source code on the Earth System Grid (ESG) website. See below for detailed instructions.

In versions 5.0.0 and later, NCL and NCAR Graphics are now available as one package and referred to as "NCL".

To download and install NCL on your system, you need to:

- Request an account from the Earth System Grid website (a one-time deal)
- Download the appropriate NCL binaries for your system, or the source code
- Install the NCL binaries, or build and install NCL from source code.
- Here is a troubleshooting guide for building NCL from source code.
- Optionally download the RANGS/GSHHS database if you need access to high- resolution map coastlines.

For more information, visit the following website.
http://www.ncl.ucar.edu/Download/index.shtml

(2) Set up NCL Environment

Before you use NCL, you must do four things:

① Set the NCARG_ROOT environment variable.
② Add $NCARG_ROOT/bin to your UNIX search path.
③ Make sure your DISPLAY environment variable is set correctly.
④ Put a .hluresfile file in your home directory.
⑤ Source vapor-setup.csh in the shell before you run NCL. This is needed to run the wrf2geotiff command.

For more information, visit the following website.
http://www.ncl.ucar.edu/get_started.shtml#SetUpEnvironment

(3) Run NCL scripts

Once your NCL environment has been correctly set up, you are ready to run NCL. If you already have an NCL script, say " wrf_Acc_Precip.ncl", you can run it by typing:

```
% ncl wrf_Acc_Precip.ncl
```

References

1. VAPOR General Information

VAPOR Home Website http://www.vapor.ucar.edu

2. VAPOR Documentation

- VAPOR Quick Start Guide
- Getting started with VAPOR and WRF
- VAPOR/WRF Data and Image Preparation Guide
- Visualization of WRF Data using VAPOR: A Georgia Weather Case Study

Documents Download Website: http://www.vapor.ucar.edu

Special Thanks to

- Dr. Bill Kuo (ESSL/NCAR)

  for advising on the choice of Case Study Topics

- Dr. Wei Wang (ESSL/NCAR)

  for providing a WRF Model Data of Typhoon Jangmi

- Dr. Song You Hong (DAS/Yonsei University)

  for advising on the variables to visualize Typhoon

- Dr. Alan Norton (CISL/NCAR)

  for advising on the VAPOR usage

- Mr. Rick Brownrigg (CISL/NCAR)

  for preparing a document about georeferenced imagery