

McIDAS-V Tutorial

Basic and Advanced Training

updated September 2019 (software version 1.8)

McIDAS-V is a free, open source, visualization and data analysis software package that is the next generation in SSEC's 40-year history of sophisticated McIDAS software packages. McIDAS-V displays weather satellite (including hyperspectral) and other geophysical data in 2- and 3-dimensions. McIDAS-V can also analyze and manipulate the data with its powerful mathematical functions. McIDAS-V is built on SSEC's VisAD and Unidata's IDV libraries. The functionality of SSEC's HYDRA software package is also being integrated into McIDAS-V for viewing and analyzing hyperspectral satellite data.

More training materials are available on the McIDAS-V webpage and in the Getting Started chapter of the McIDAS-V User's Guide, which is available from the Help menu within McIDAS-V. You will be notified at the startup of McIDAS-V when new versions are available on the McIDAS-V webpage - <http://www.ssec.wisc.edu/mcidas/software/v/>.

If you encounter an error or would like to request an enhancement, please post it to the McIDAS-V Support Forums - <https://mcidas.ssec.wisc.edu/forums/>. The forums also provide the opportunity to share information with other users.

In this McIDAS-V Tutorial, some exercises will be explained using different methods of data access: local data files, pre-loaded data bundles and real-time access to default remote servers. If you have access to your own real-time ADDE servers, you may also use those, but be aware that different server configurations may make the explanations in this document not quite applicable to all data that you may load.

Terminology

There are two windows displayed when McIDAS-V first starts, the **McIDAS-V Main Display** (hereafter **Main Display**) and the **McIDAS-V Data Explorer** (hereafter **Data Explorer**).

The **Data Explorer** contains three tabs that appear in bold italics throughout this document: ***Data Sources***, ***Field Selector***, and ***Layer Controls***. Data is selected in the ***Data Sources*** tab, loaded into the ***Field Selector***, displayed in the **Main Display**, and output is formatted in the ***Layer Controls***.

Menu trees will be listed as a series (e.g. ***Edit -> Remove -> All Layers and Data Sources***).

Mouse clicks will be listed as combinations (e.g. ***Shift+Left Click+Drag***).

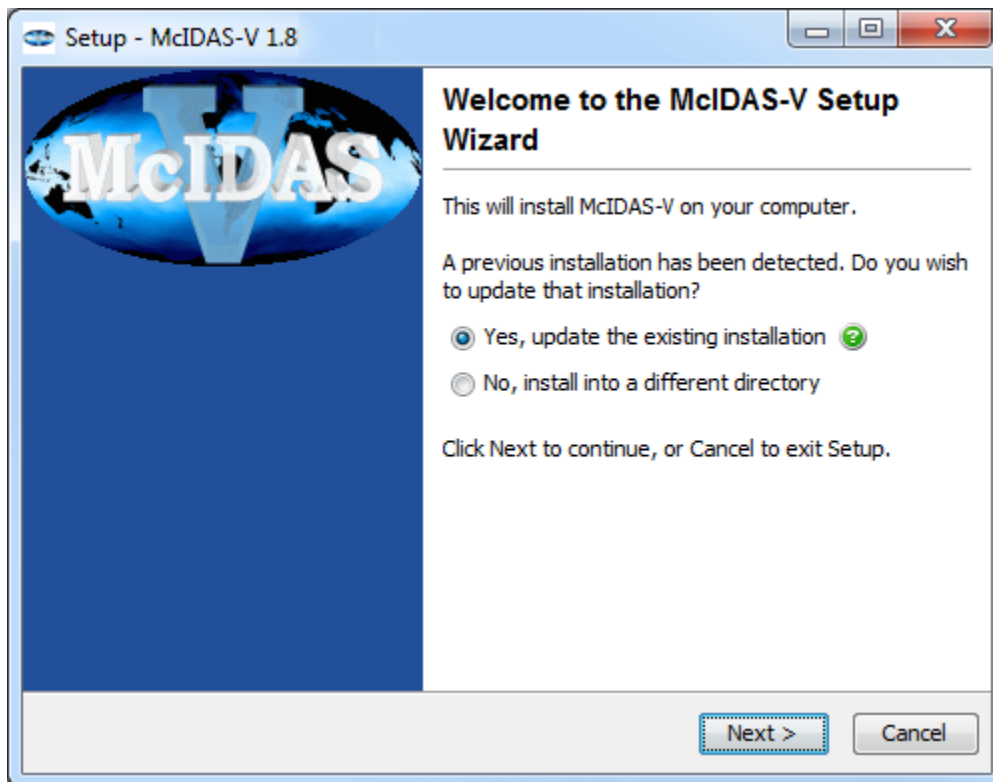
Introduction

Installation of McIDAS-V

The installation of McIDAS-V is a simple three step procedure (download, install, and run).

1. Download the appropriate package for your operating system from the McIDAS-V webpage. This file is only the installer and can be placed anywhere on your machine. When you run the installer in the next step, you can then indicate where you want McIDAS-V to be installed. The system requirements for McIDAS-V are also listed on the McIDAS-V webpage.
2. Install McIDAS-V by starting the installer by following the instructions appropriate for your operating system:

macOS	mount the .dmg and double-click the installer
Windows	double-click the downloaded .exe file
All other platforms	open a terminal window and run sh ./<installer>.sh



A GUI will walk you through the installation steps and allow you to create a program group and/or desktop icon.

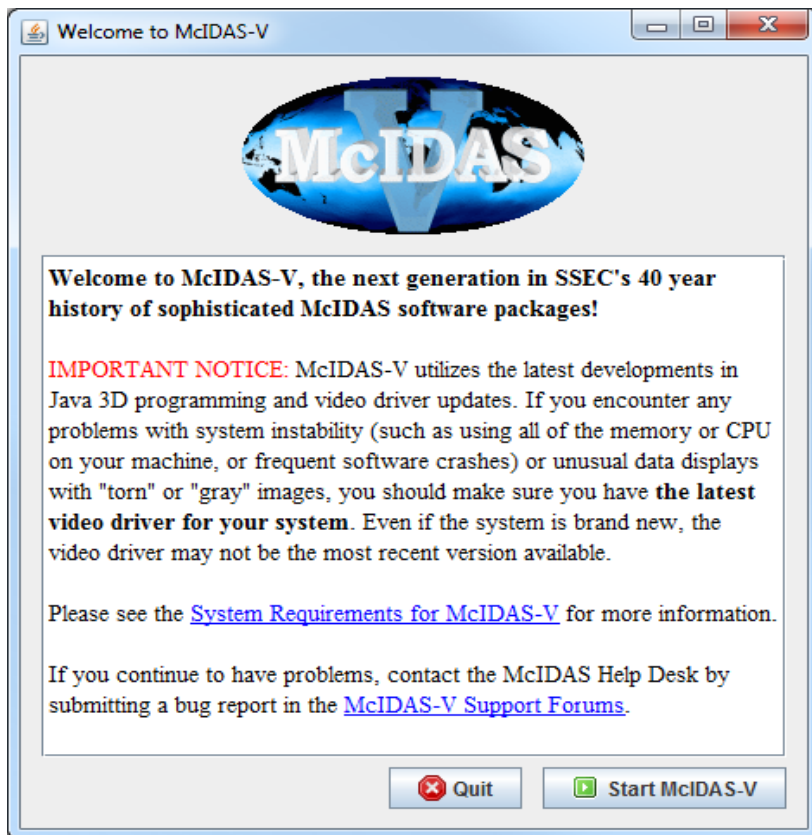
3. Run McIDAS-V by following the instructions appropriate for your operating system:

macOS	<i>Double-click</i> on the McIDAS-V shortcut icon that was created in /Applications.
Windows	<i>Double-click</i> on the McIDAS-V shortcut icon that was created on the Desktop.
All others	At the UNIX prompt from the directory where McIDAS-V was installed, run the command: runMcV

Note: By default, McIDAS-V uses 80% of the available memory on your machine. The maximum amount of memory is determined by the operating system. To manually change the amount of memory used by McIDAS-V, edit the **Maximum Heap Size** in the **Advanced** tab of the Preferences by selecting **Edit -> Preferences...** from the main menu. The new amount of memory will be saved and used in subsequent sessions. For 32 bit operating systems, it is recommended to set this to no more than 1250 MB. The maximum value for 32 bit operating systems is 1536 MB. Exit McIDAS-V and start a new session to use this new memory setting.

Welcome Window

Upon initial startup, the **Welcome to McIDAS-V** window will be displayed.

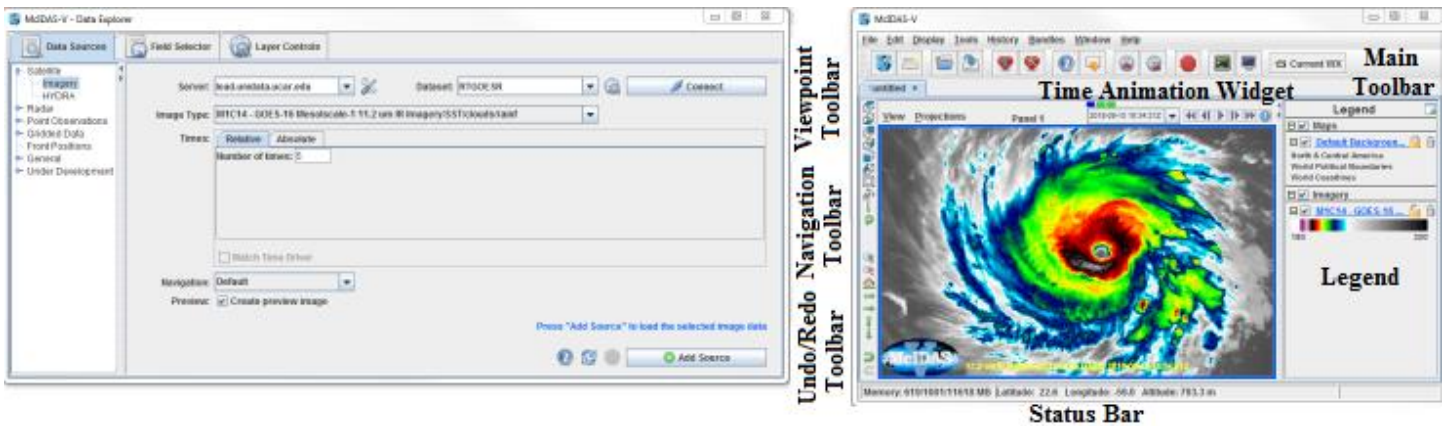


Please be sure to read this **IMPORTANT NOTICE**. Because McIDAS-V utilizes the latest developments in Java 3D programming and video driver updates, it is often necessary to update video drivers to solve some seemingly unrelated performance problems. Do not assume that because you have a new machine, you have the latest drivers. We have found brand new machines with drivers that are more than five years old!

After reading the notice in the Welcome window, click **Quit** to close the window, or click **Start McIDAS-V** to close the window and start McIDAS-V. Once McIDAS-V has been started, this window will not appear again.

Introduction to McIDAS-V windows

There are two main windows in the McIDAS-V application - the **Data Explorer** window and the **Main Display** window. Other windows may appear when needed.



Data Explorer (shown on the left)

The **Data Explorer** window is central to McIDAS-V. It is used to choose data sources and parameters to display, the kinds of displays to make, and times of data to display. The first time McIDAS-V starts up, the **Data Explorer** defaults to the *Satellite -> Imagery Chooser*. This chooser, along with several others, uses ADDE (Abstract Data Distribution Environment) servers to access data. McIDAS-V comes with a pre-configured list of several available free servers which contain real-time and archived atmospheric datasets.

Main Display (shown on the right)

The **Main Display** window includes the McIDAS-V display panels, legend, time animation controls, viewpoint controls for 3D displays, icons to zoom and pan, menus to control views and projections, and the main McIDAS-V menu. Navigating to *Help -> User's Guide* in the Main Toolbar brings up the McIDAS-V User's Guide, which has both a table of contents and a search function.

Common Usage Scenario

To create displays of data with McIDAS-V, the common usage scenario is:

- Choosing the source of data (local files, remote servers, etc.)
- Selecting parameters and display type
- Creating the display
- Controlling the display
- Creating a bundle

Bundles

Bundles let you save a quick "snapshot" of McIDAS-V, including data sources, maps, and data displays. Bundles are small information files that specify the state of McIDAS-V. They are a kind of configuration file. They include information about what data sources are in use, which parameters from the data sources are displayed, and how they are displayed.

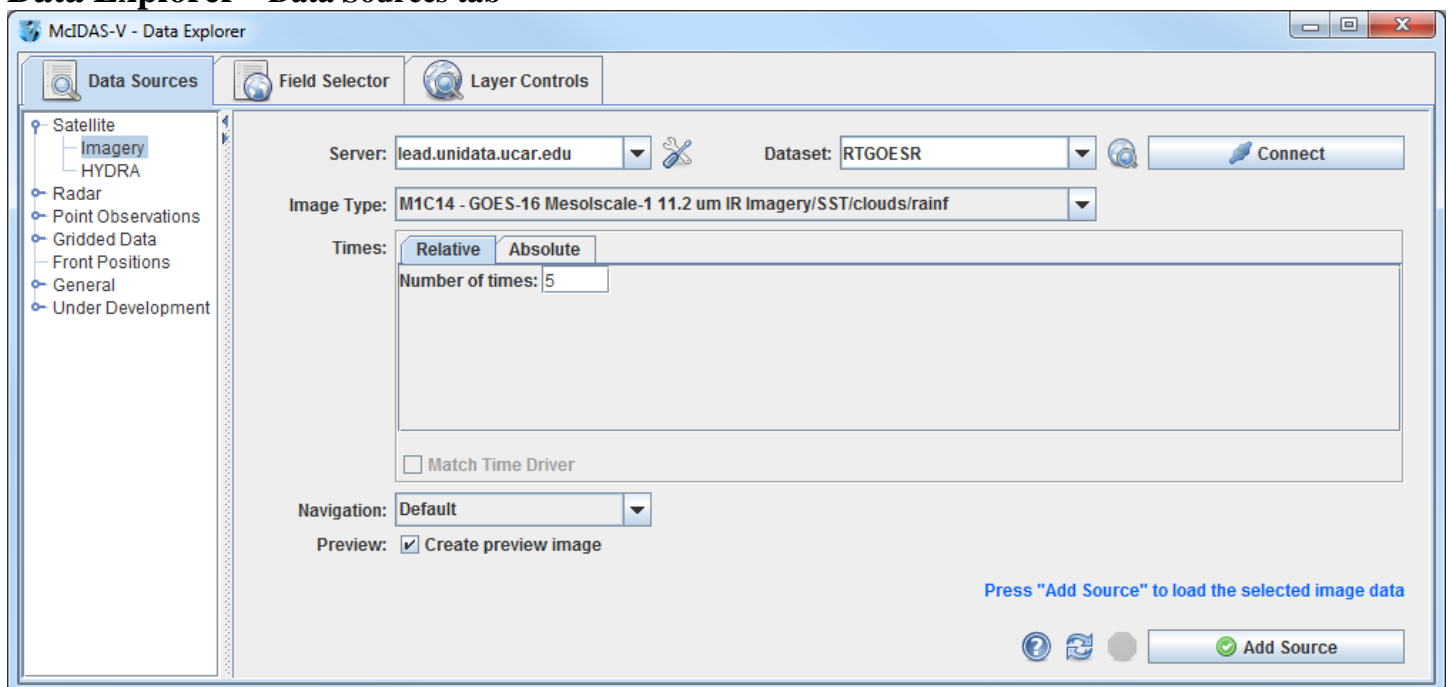
The purpose of bundles is for you to save a particular McIDAS-V setup and display. A bundle can be used for your own reference at a later time, or others using McIDAS-V can use bundles you made to see data the same way you did. There are two types of bundles, regular *.mcf bundles, and *.mcfv zipped bundles. These serve different purposes.

The *.mcf bundle, along with storing the setup and display options, stores a reference to the particular type of data you have loaded into McIDAS-V. For example, if you load in the most recent GFS, display pressure, and

save your display as a *.mcv bundle, the next time the bundle is loaded, McIDAS-V will connect to the server/catalog where you got the data from originally, and display the most recent GFS pressure data. This type of bundle can be very useful in an operational environment, where you often want to look at the same products displaying the most recent data.

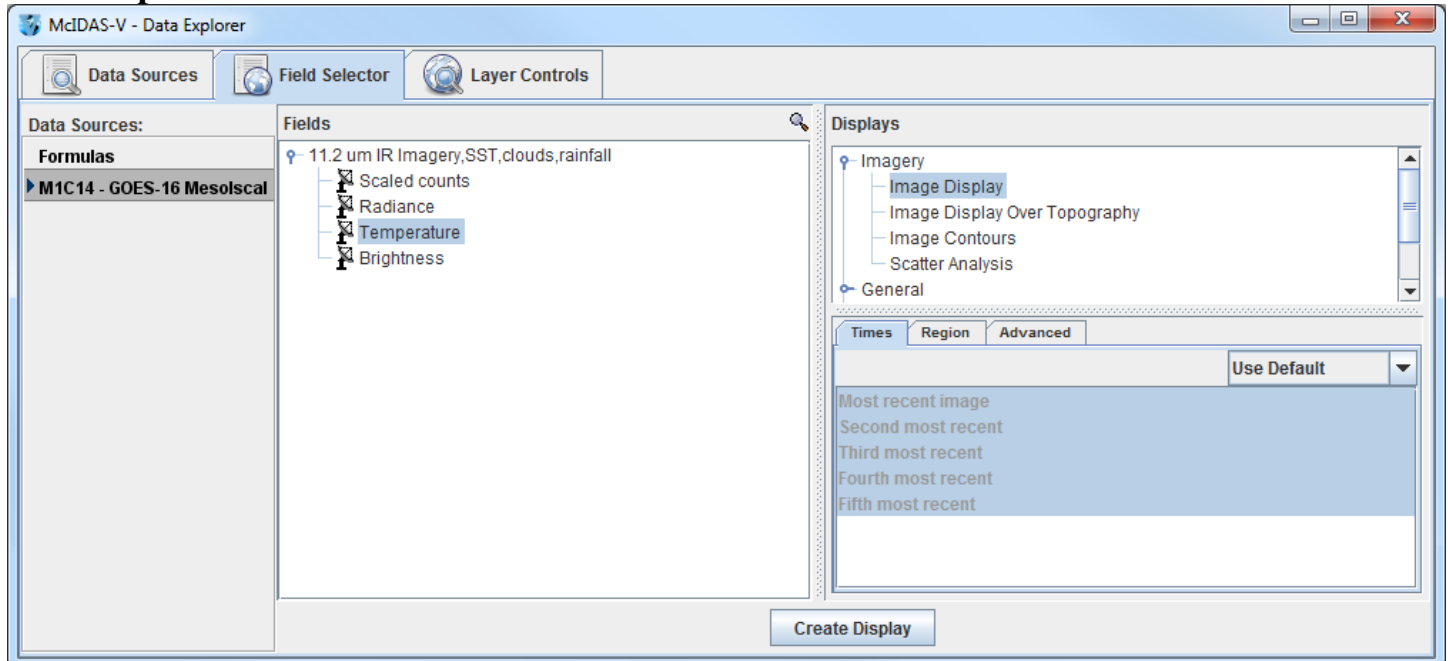
The *.mcvz bundle is different from the *.mcv bundle in that it actually stores data along with the bundle. For example, if you load in radar data for a tornado case that you want to look at again, but that may not be on the server at a later date, you can save your display as a *.mcvz zipped bundle. The next time this bundle is loaded in, the same exact data with the same display options will be displayed. This can be useful for case studies, where you always want to look at the same data.

Data Explorer - Data Sources tab



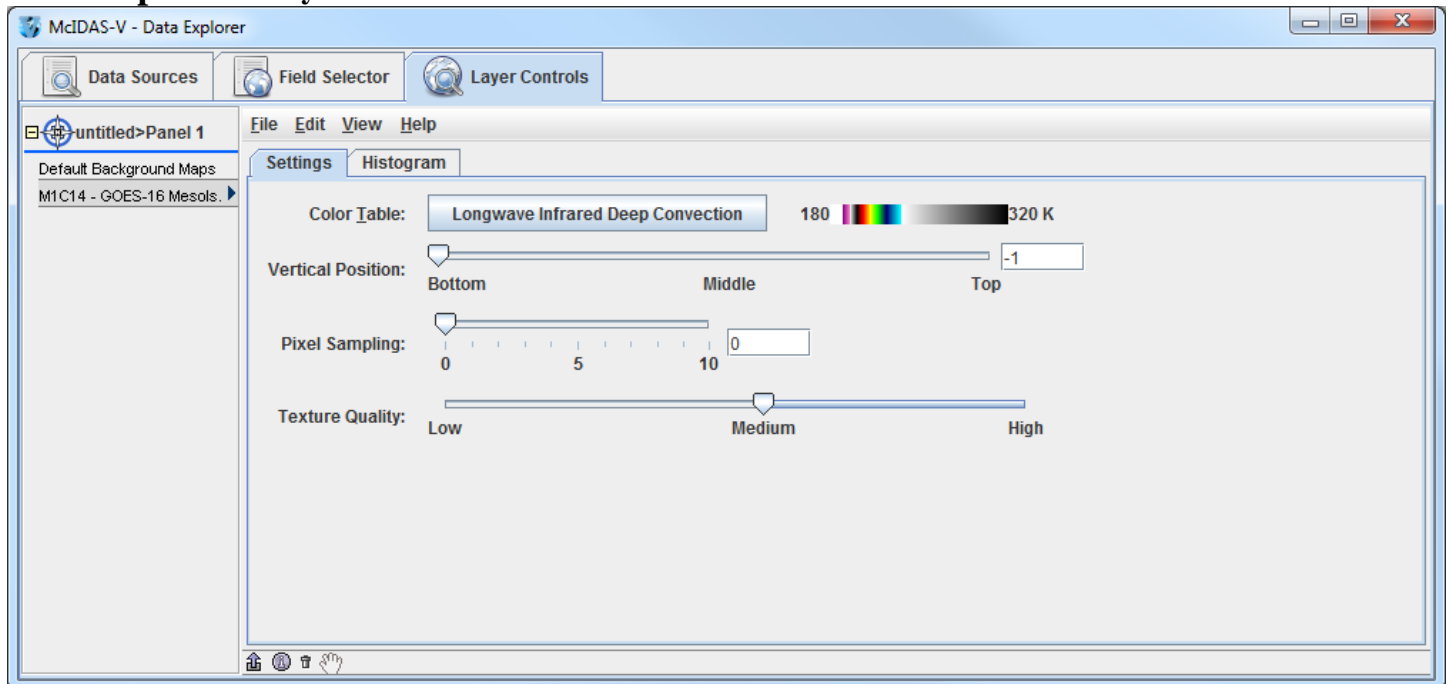
The **Data Sources** tab is used to locate remote data servers or local files for display in McIDAS-V. Once the type of data source is selected, a chooser with the appropriate selection criteria is displayed. **Satellite -> Imagery** is chosen above, and the **Satellite -> Imagery** chooser is displayed. Other data source types are the HYDRA multi-spectral display, Radar, Point Observations, Gridded Data and General files.

Data Explorer - Field Selector tab



The **Field Selector** tab analyzes the data source to determine the displayable fields and parameters. With this information, a list of available displays is presented. In the **Subset** panel of the **Field Selector** there are tabs to select which times to display, which region to use, as well as a variety of advanced options. Note that the tabs in the **Subset** panel will differ dependent on which data source you loaded.

Data Explorer - Layer Controls tab



The **Layer Controls** tab contains the display options for the displayed fields and parameters which are referred to as layers. Changes made to each layer will be reflected in the **Main Display**. Also, if a graph, time series or special display was selected, it will display in the **Layer Controls**.

Introduction to McIDAS-V controls

1. Load the *<local path>/Data/Installation/Intro-Bundle.mcvz* bundle to display NCEP GFS data in the **Main Display** window.
 - a. In the **Main Menu Bar** in the **Main Display** window of McIDAS-V, select *File -> Open File....*
 - b. In the file browser, “Look In” the location where you downloaded the data files for this tutorial, select **Intro-Bundle.mcvz** and click **Open**.
 - c. When prompted, select *Replace Session* to replace the current window with the contents of the bundle and click **OK**.
 - d. Select *Remove all layers*.
 - e. Select *Remove all data*.
 - f. Select *Write to temporary directory*, check *Don't show this again*, and click **OK**.

The **Main Display** will change to a view from the south, and two new 3D Surfaces will be listed in the **Legend** in the right panel of the **Main Display** window. One surface is a color-shaded image over topography. The other surface is an isosurface of a constant wind speed of 40 m/s. The green isosurface is shadowed for perspective, otherwise it would be one block of color. Additionally, a 3D map is listed in the legend, allowing the map lines to appear on top of the topographic data.

2. Use *Right-Click+Drag* to rotate the display. Use the navigation controls listed on the last page of this document or the navigation controls on the left side of the **Main Display** to zoom in and out, and “move through” the data.
3. Toggle the 3D surfaces on and off in the display by clicking the checkbox next to each item in the **Legend** on the right side of the **Main Display** window.
4. Click on the name of each item in the **Legend** to bring up the *Layer Controls* to change the color table or the isosurface values. To change the unit of the isosurface to something other than **m/s**, in the *Layer Controls* menu select *Edit -> Change Display Unit*.
5. Use the **Time Animation** controls at the top of the **Main Display** to loop through the images, or step through them one by one.

Below is a list of supported data types, formats, and the method to access them in McIDAS-V.

Data Type	Description	Supported Formats	Access Method
Gridded	Numerical weather prediction models, climate analysis, gridded oceanographic datasets, NCEP/NCAR Reanalysis	netCDF	local files, HTTP, TDS Servers
		GRIB (versions 1&2)	local files, TDS Servers
		Vis5D	local files, HTTP
		GEMPAK	local files, TDS Servers
Satellite Imagery	Geostationary and polar orbiter satellite imagery, derived satellite products	ADDE	ADDE servers, local & remote
		McIDAS AREA	local files, local & remote ADDE
		AIRS	local files
		GINI	local files, TDS servers
		AMSR-E Level 1b	local ADDE
		AMSR-E Level 2a	local ADDE
		AMSR-E Rain Product	local ADDE
		EUMETCast LRIT	local ADDE
		Meteosat OpenMTP	local ADDE
		Meteosat Second Generation (MSG) Level 1b	local ADDE
		Metop AVHRR Level 1b	local ADDE
		MODIS L1b MOD02 (MODIS Level 1b)	local ADDE
		MODIS L2 MOD04 (Level 2 Aerosol)	local ADDE
		MODIS L2 MOD06 (Level 2 Cloud Top Properties)	local ADDE
		MODIS L2 MOD07 (Level 2 Atmospheric Profile)	local ADDE
		MODIS L2 MOD28 (Level 2 Sea Surface Temperature Products)	local ADDE
		MODIS L2 MOD35 (Level 2 Cloud Mask)	local ADDE
		MODIS L2 MODR (Level 2 Corrected Reflectance)	local ADDE
		MSG HRIT FD and HRV	local ADDE
		MTSAT HRIT	local ADDE
		NOAA AVHRR Level 1b	local ADDE
		SSMI (TeraScan netCDF)	local ADDE
		TRMM (TeraScan netCDF)	local ADDE
		Himawari-8 (*.DAT) *	local ADDE
		INSAT-3D Imager *	local ADDE
		INSAT-3D Sounder *	local ADDE
		GOES-16 ABI (Level 1b) *	local ADDE

Data Type	Description	Supported Formats	Access Method
Radar	Radar images	Level II	local files or TDS (bzip2 compressed or uncompressed)
		Level III/TDWR	ADDE Servers, local files, or TDS
		Universal Format (UF)	local files
		DORADE	local files
Point Observational	Surface observations (METAR and SYNOP), earthquake observations	ADDE	ADDE servers
		netCDF (Unidata, AWIPS/MADIS formats)	local files
		Text (ASCII, CSV), Excel spreadsheet	local files
	Global balloon soundings (RAOB)	ADDE	ADDE servers
		netCDF (Unidata, AWIPS/MADIS formats)	local files
		CMA text format	local files
Trajectory	Aircraft observations	netCDF (RAF convention)	local files
		Text (ASCII, CSV)	local files
GIS	Data typically used in Geographic Information Systems (GIS)	ESRI Shapefile	local files, HTTP
		USGS DEM	local files
QuickTime	QuickTime movies (without extensions)	QuickTime	local files, HTTP
Flat Files	Flat (2-dimensional) data	Binary, ASCII, standard images (JPEG, GIF, etc.)	local files; Flat files chooser


* The Himawari-8, INSAT-3D, and ABI local servers are currently supported on Linux and OS X platforms

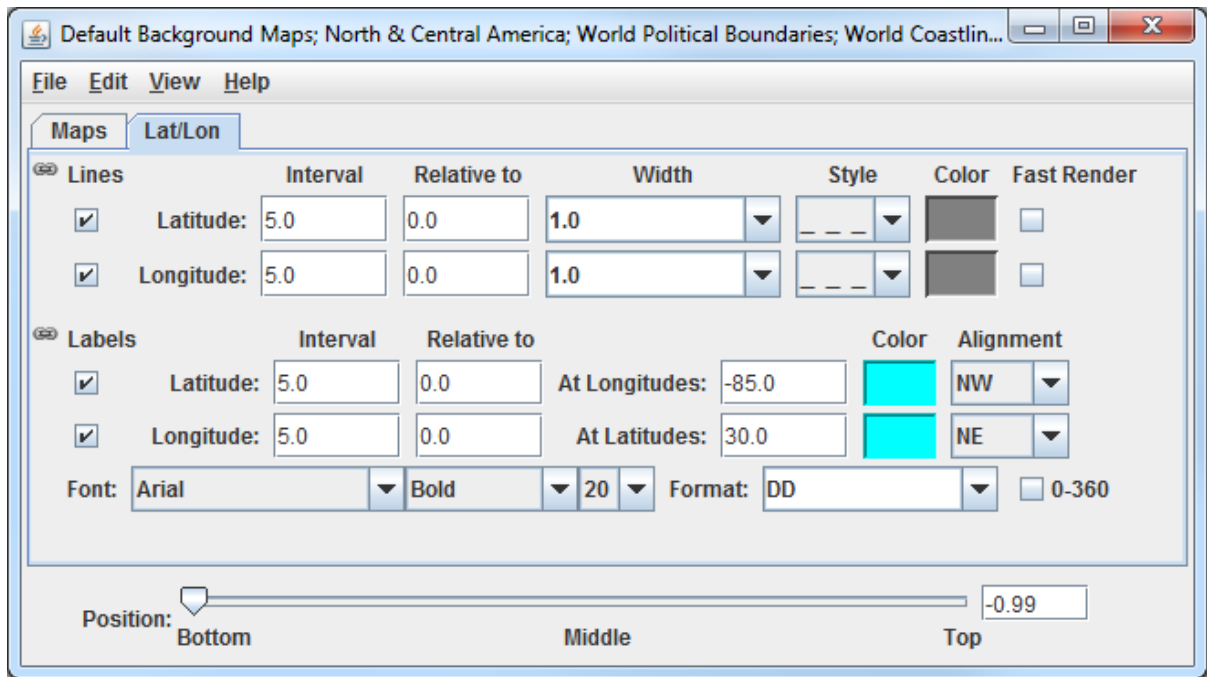
Geostationary Satellite Data

1. In the **Data Sources** tab of the **Data Explorer**, navigate to the **Satellite -> Imagery** chooser. Select *atm.ucar.edu* as the **Server** and *RTGOESR* as the **Dataset**. Click **Connect**.
2. Select *FD – GOES-16 FD all bands* as the **Image Type**.
3. In the **Times** panel, select the **Relative** tab. For **Number of times**, enter *1* and click **Add Source** to create a data source of the most recent time.
4. Display band 13 temperature data.
 - a. Navigate to the **Field Selector** tab of the **Data Explorer** to see a listing of each band included with the data. Select the *10.3 um IR Surface & cloud -> Temperature* field.
 - b. From the **Displays** panel, select the **Imagery -> Image Display** display type.
 - c. Select the **Region** tab to see a low-resolution preview image of the data. Note that there is a green box drawn around the preview image. Any data inside this bounding box will be displayed.
 - d. Select the **Advanced** tab to see information about the band and calibration selected. These values match the domain of the data selected in the **Region** tab, so it covers the entire domain of the data. Notice that the **Image Size** is only *904 x 904* when the raw size of the data is *5424 x 5424*. The magnification of the data is set to *-6, -6*. This reduction in resolution is the default to allow for an efficient display of the data. Modify the **Advanced** tab to display full resolution data over Florida.
 - i. **Coordinate Type:** *Latitude/Longitude*
 - ii. **Location:** *Center*
 - iii. **Lat:** *28*; **Lon:** *-83.5*
 - iv. **Image Size:** *400 x 400*
 - v. **Magnification:** *1 x 1*
 - e. Click **Create Display**.
5. Investigate the display.
 - a. In the **Main Display** window, practice navigating around in the display. To do this, use *Control+Right-Click+Drag* to translate the display. This shifts the data left/right and up/down without rotating the display. Zoom in on the display by scrolling down on the mouse wheel. As an alternative, select a region in the **Main Display** by using *Shift+Left-Click+Drag* to zoom in on a specific domain.
 - b. Probe individual pixel values by holding down the middle mouse button over the display. This will print out probe values at the bottom of the display. Notice that latitude, longitude, and temperature values are included in the probe output.

- c. Navigate to the **Histogram** tab of the **Layer Controls** to see a histogram of band 13 temperature values.
6. Add a data transect to the display.
 - a. From the **Field Selector** tab of the **Data Explorer**, select the **Data Transect** display type and click **Create Display**.
 - b. In the **Main Display** window, there is a red line that is referred to as the data transect. The values along this transect line are output in the **Layer Controls** tab of the **Data Explorer**. The two end points of the transect can be moved around by *Left-Clicking+Dragging* them, or by entering new Lat/Lon values at the bottom of the **Display** tab in the **Layer Controls**. The transect line as a whole can be moved by *Left-Clicking+Dragging* the triangle in the middle of the transect line.
 7. Create a display of the 10.3 micron temperatures so only values of 273K and less are plotted.
 - a. Add a new two-paneled map display by selecting **File -> New Display Tab -> Map Display -> Two Panels** menu item.
 - b. *Left-Click* in the left panel to make this panel active. The active panel has a blue frame around it, and this is the panel any displays will be added to.
 - c. In the **Field Selector** tab of the **Data Explorer**, select the **10.3 um IR Surface & cloud -> Temperature** field. Choose the **Imagery -> Image Display** display type and click **Create Display**.
 - d. Create the cloud mask display.
 - i. *Left-Click* in the right panel to make this panel active. The active panel has a blue frame around it, and this is the panel any displays will be added to.
 - ii. In the **Field Selector** tab of the **Data Explorer**, select **Formulas**.
 - iii. Select the **Imagery -> Mask Value** formula, the **Imagery -> Image Display** display type, and click **Create Display**.
 - iv. In the **Select input** window, enter the following and then click **OK**:
 1. **comparison:** <
 2. **cutoff:** 273
 3. **useNaN:** 1

Notes: The comparison is set to < or 'less than' since the goal is to display values less than 273, which was set as the cutoff value. Setting useNaN to 1 makes it so that if the pixels were probed outside of the masked range (where values are greater than 273) they wouldn't show a value. useNaN can be set to 0 as well, and in this case pixel values greater than 273 would probe as 0. Note that if these pixel values are set to 0, then these 0 values will be included in any calculations with the data that may be done later.

- v. In the new **Field Selector** window, for **inputFieldForMask**, select *10.3 um IR Surface & cloud -> Temperature*. This is the field that is being masked.
 - vi. For **displayFieldToBeMasked**, select *10.3 um IR Surface & cloud -> Temperature*. This is the field that will be displayed where the **inputFieldForMask** is less than 273 degrees. In this case, we are displaying the same field that is being masked.
 - vii. Click **OK**.
- e. Investigate the display. Note that as one of the panels is navigated, the other will be navigated in the same way. This is because by default panels in the same tab are set to share views. This setting can be disabled in a panel by unchecking *Projections -> Share Views* in the display panels.
8. Create a display of a subtraction of two bands.
- a. Add a new one-paneled map display by selecting *File -> New Display Tab -> Map Display -> One Panel* menu item, or by clicking the **Add new tab** () button in the **Main Toolbar**.
 - b. In the *Field Selector* tab of the **Data Explorer**, select **Formulas**.
 - c. Select the *Miscellaneous -> Simple difference a-b* formula, the *Imagery -> Image Display* display type, and click **Create Display**.
 - d. In the new **Field Selector** window, for **a**, select the *8.44 um IR Total WV cloud phase, dust -> Temperature* field. For field **b**, select the *7.34 um IR Lower-level WV, winds & SO2-> Temperature* field.
 - e. Click **OK** in the **Field Selector** window to display the band subtracted data.
 - f. *Right-Click* on the colorbar of the ABI layer in the **Legend** and select the *System -> Inverse Gray Scale* enhancement.
 - g. Add latitude and longitude labels to the display.
 - i. *Left-Click* on the blue text for the *Default Background Maps* layer in the **Legend** to get to the **Layer Controls** tab of the **Data Explorer** for the layer.
 - ii. From the *Lat/Lon* tab of the **Layer Controls**, set the following:
 1. Check the visibility checkboxes for **Lines** and **Labels** to plot them in the display.
 2. **Lines: Interval: 5.0**
 3. **Labels: Interval: 5.0; At Longitudes: -85.0, At Latitudes: 30.0, Color: light blue**
 4. **Font: Arial; Bold; 20**



- h. Investigate the display to see the latitude and longitude lines and labels. Probe the data to see the temperature difference between the two bands.
9. Capture an image of the display.
 - a. From the **Main Display**, select the **View -> Capture -> Image** menu item.
 - b. In the **Save As** text field, enter the name to give your image, for example: *band_subtraction.gif*.
 - c. Use the dropdown below **Save As** to select the directory to save your image to.
 - d. Under **Capture What**, select *Current View*. This will capture an image of the display without including any other components of the **Main Display** such as menu items and the **Legend**.
 - e. Click **Save**.
 - f. View your saved image to verify the image was captured correctly.

Accessing GOES-16 ABI Data from a Remote Server – 5 Timesteps

10. Remove all layers and data sources from any previous displays via the **Edit -> Remove -> All Layers and Data Sources** menu item from the **Main Display**.
11. Add a new one-paneled map display by selecting **File -> New Display Tab -> Map Display -> One Panel** menu item. Close any previously-existing tabs.
12. In the **Data Sources** tab of the **Data Explorer**, navigate to the **Satellite -> Imagery** chooser. In the **Relative** tab of the **Times** panel, for **Number of times**, enter 5 and click **Add Source**.

13. Display a loop of 10.4um Temperature data over Wisconsin and Illinois.

- a. Navigate to the **Field Selector** tab of the **Data Explorer** to see a listing of each band included with the data. Select the **10.3 um IR Surface & cloud -> Temperature** field.
- b. From the **Displays** panel, select the **Imagery -> Image Display** display type.
- c. Navigate to the **Advanced** tab, and enter the following information:
 - i. **Coordinate Type:** *Latitude/Longitude*
 - ii. **Location:** *Center*
 - iii. **Lat:** *42*; **Lon:** *-89.6*
 - iv. **Image Size:** *450 x 450*
 - v. **Magnification:** *1 x 1*
- d. Click **Create Display**.

14. Investigate the display.

- a. The **Main Display** now has a loop of the 5 most recent times of data in the FD dataset. As done in step 5 earlier, navigate around and zoom in/out of the display. Probe the display by holding down the middle mouse button over the data.
- b. Use the Time Animation Widget at the top of the display panel to play through a loop of the data.



- c. Click the **Properties** button (i) in the Time Animation Widget to modify settings of the animation including the dwell rate (the time spent on each frame in the animation). As an example, set the **Forward, First, and Last** dwell rates to *0.2* and click **OK**. This reduced dwell rate allows the display to loop faster.

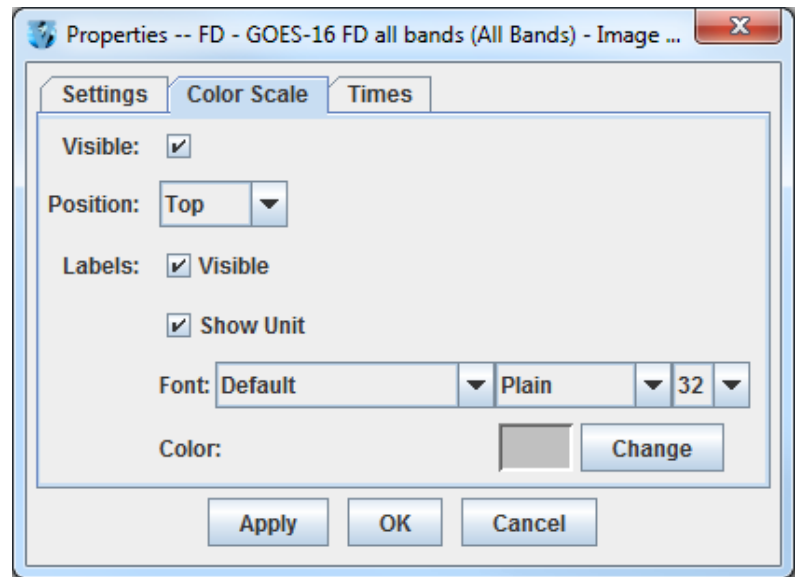
15. Change the enhancement used in the display.

- a. Stop the animation by clicking the **Stop Animation** button in the Time Animation Widget (II).
- b. *Right-Click* on the colorbar of the ABI layer in the **Legend** and select the **Satellite -> GOES-R -> ABI IR Temperature** enhancement.

16. Add a color scale to the display.

- a. *Right-Click* on the ABI layer in the **Legend** and select **Edit -> Properties**.

- b. In the **Properties** window, navigate to the **Color Scale** tab and set the following:
 - i. **Visible:** Check this option to add the color scale to the display.
 - ii. **Position:** *Top*. This will position the color scale at the top of the display.
 - iii. **Labels -> Visible:** Check this option to add numerical labels to the color scale.
 - iv. **Labels -> Show Unit:** Check this to add the display unit to the color scale labels.
 - v. **Labels -> Font:** Set the size to 32.



- c. Click **OK**.

17. Create a Data Probe/Time Series display of band 13 temperature data. This display type allows for plotting how temperature values at a specific location change through time throughout the loop.

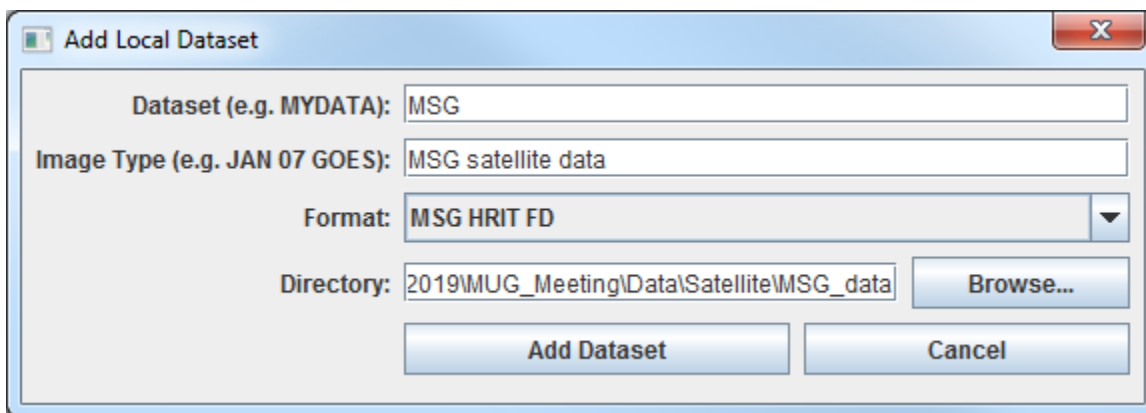
- a. In the **Field Selector** tab of the **Data Explorer**, select the **Data Probe/Time Series** display type and click **Create Display**.
- b. The **Layer Controls** tab of the **Data Explorer** now shows a time series display of temperature values at the location of the probe in the **Main Display**. Change the range of the y-axis (temperature values) to match the range of the data.
 - i. At the bottom of the **Layer Controls** tab, *Double-Click* on the *186-Band13_TEMP* parameter to open the **Properties** window.
 - ii. In the **Properties** window, change the **Range** text boxes to a **Min** of 180 and a **Max** of 320 to match the range of the enhancement in the **Main Display**.
 - iii. Click **OK** to apply the new range in the **Layer Controls** and close the **Properties** window.
- c. Move the data probe in the **Main Display** around with *Left-Click+Drag*. The time series display in the **Layer Controls** automatically updates itself as the probe's location changes.

The data probe in the **Main Display** can also be set to a specific latitude/longitude value by using the **Lat** and **Lon** text boxes at the bottom of the **Layer Controls**. For example, enter values of 38.6 for **Lat** and -88.1 for **Lon**. Note that you must press *Enter* in each text field for latitude/longitude to update the location of the data probe.

Using a Local ADDE Dataset


18. Create a local dataset to access the imagery files on your local machine. If you want to load real-time data, skip to step 3.

- a. In the **Main Menu Bar** of the **Main Display**, select *Tools -> Manage ADDE Datasets*.
- b. In the **ADDE Data Manager**, select *File -> New Local Dataset*. Enter the following parameters to set up a dataset with the AREA files provided with this tutorial:
 - **Dataset** – *MSG*
 - **Image Type** – *MSG satellite data*
 - **Format** – *MSG HRIT FD*
 - **Directory** – select the *<local path>/Data/Satellite/MSG_data* directory



- c. Click **Add Dataset**. Close the **ADDE Data Manager** by clicking **Ok** or selecting *File -> Close*.

19. To load the data in this local dataset, follow these steps.

- a. Click on the  button in the main toolbar to go to the **Data Explorer**.
- b. From the *Data Sources* tab, select the *Satellite -> Imagery* chooser.
- c. Select *<LOCAL-DATA>* for **Server**, select *MSG* for **Dataset**, click **Connect**.
- d. Choose the *MSG satellite data* **Image Type** and select an **absolute** time of *10:00 UTC*.
- e. Click **Add Source** to show the *Field Selector*.
- f. In the **Fields** panel of the *Field Selector*, choose *0.6 um VIS Cloud and Surface Features -> Brightness*.

g. In the **Advanced** tab, set the following values:

- a. Coordinate Type: *Latitude/Longitude*
- b. Location: *Center*
- c. Lat: *16.3; 84.7*
- d. Image Size: *752 x 616*
- e. Line Mag: *1*
- f. Ele Mag: *1*

h. Click **Create Display**. The 12:00 UTC 0.6 μm is displayed in the **Main Display** window. The image displays Cyclone Fani, which was the equivalent of a category 4 hurricane at the time of this image.

The screenshot shows the 'Advanced' tab of a software interface. It contains the following settings:

- Coordinate Type:** Latitude/Longitude (dropdown)
- Location:** Center (dropdown)
- Lat:** 16.3 **Lon:** 84.7 (input fields with a globe icon)
- Image Size:** 752 X 616 (input fields with a lock icon) **Raw size:** 3712 X 3712
- Magnification:**
 - Line Mag=** 1 (input field) **Res=3.0 km** (slider)
 - Ele Mag=** 1 (input field) **Res=3.0 km** (slider)
- Approx. Area:**

Center	Lat: 16.3345	Lon: 84.6797
Upper Left	Lat: 27.8336	Lon: 75.9157
Upper Right	Lat: missing	Lon: missing
Lower Left	Lat: 5.13584	Lon: 70.8662
Lower Right	Lat: 5.45700	Lon: 98.3156

Exercise

Create a 2-paneled map display. In the left panel, display the most recent GOES-16 band 1 CONUS brightness data. In the right panel, display the most recent GOES-16 band 13 CONUS temperature data. This data can be accessed from the lead.unidata.ucar.edu server, RTGOESR dataset, and CONUS image type.

Navigate around the displays to observe how the two panels move in sync with each other. Through the Projections menu of one of the display panels, disable Share Views and navigate around one of the panels again.

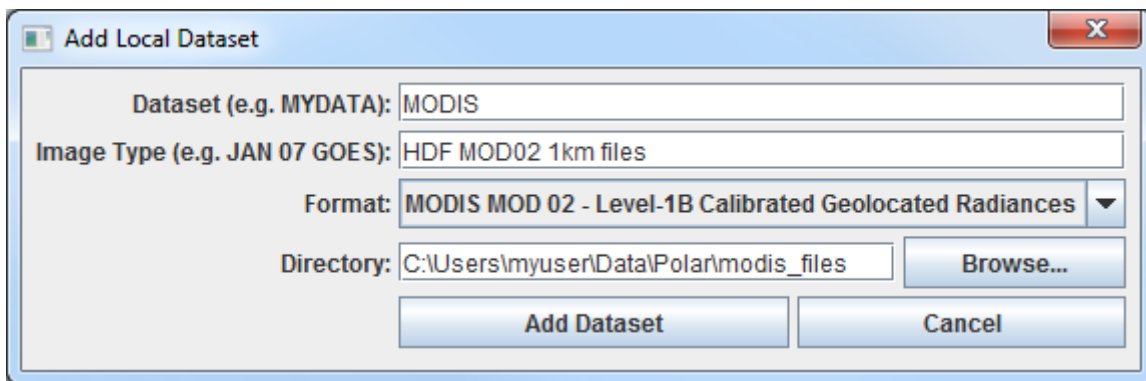
Change the name of the tab to “GOES-16 Vis and IR” by double-clicking on the “untitled” text above the panels to open the Rename Tab window.

Change the name of the left panel from “Panel 1” to “Visible” by selecting View>Properties, entering the new name into the Name field, and clicking OK. Repeat this process with the right panel to set its name to “Infrared”.


Polar Satellite Data

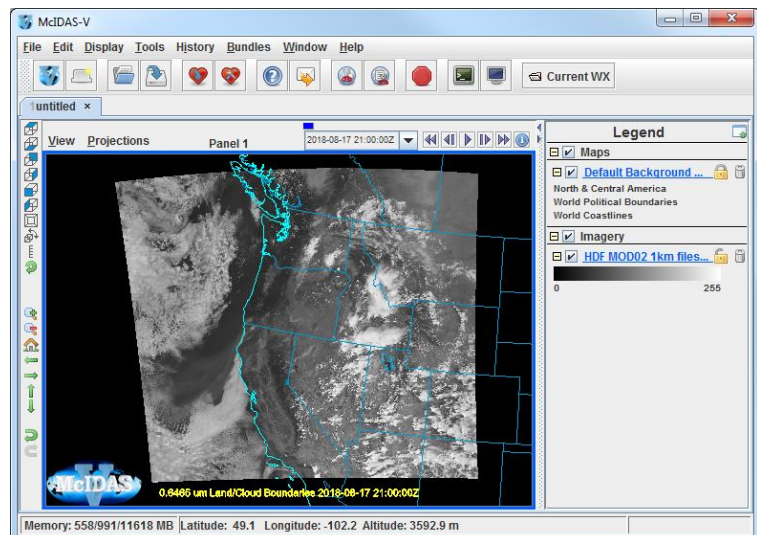
Displaying Local MODIS Data

1. Create a local dataset to access the imagery files on your local machine.
 - a. In the **Main Display** window of McIDAS-V, select **Tools -> Manage ADDE Datasets**.
 - b. In the **ADDE Data Manager**, select **File -> New Local Dataset**. Enter in the following parameters to set up a dataset with the MODIS Level 1b HDF files provided with this tutorial:
 - **Dataset** – *MODIS*
 - **Image Type** – *HDF MOD02 1km files*
 - **Format** – *MODIS MOD 02 – Level-1B Calibrated Geolocated Radiances*
 - **Directory** – select the *<local path>/Data/Polar/modis_files* directory



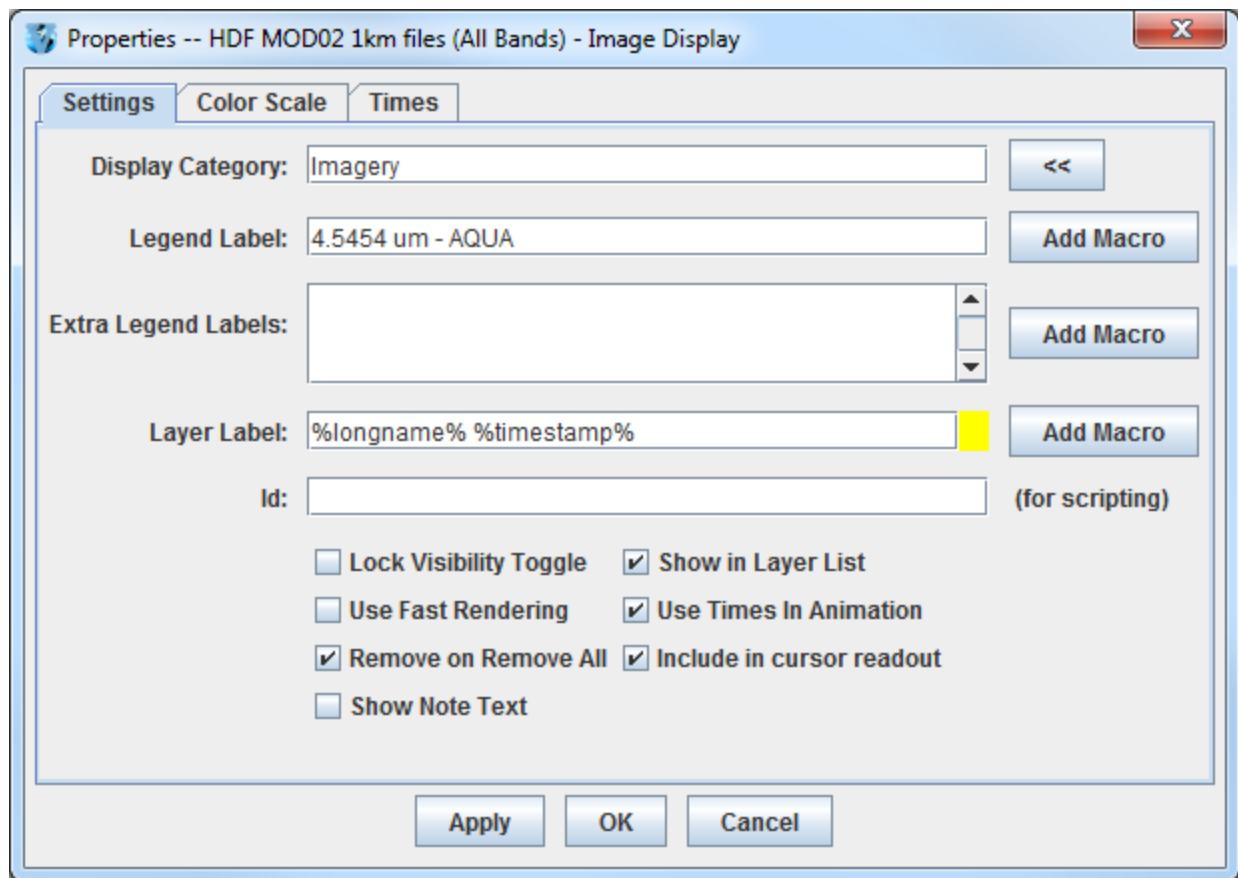
- c. Click **Add Dataset**.
 - d. Close the **ADDE Data Manager** by clicking **Ok** or **File -> Close**.
2. To load the data in this local dataset, follow these steps.

- a. Click on the  button in the **Main Toolbar** to go to the **Data Explorer**.
- b. Select the **Satellite -> Imagery** chooser from the **Data Sources** tab.
- c. Select **<LOCAL-DATA>** for the **Server**, select **MODIS** for the **Dataset**, and click **Connect**.
- d. Choose the *HDF MOD02 1km files* **Image Type** and select an **Absolute** time of *21:00 UTC*.



- e. Click **Add Source** to show the *Field Selector*.
 - f. Select **0.6465 μm Land/Cloud Boundaries -> Brightness**.
 - g. In the *Advanced*, click the green four arrow button to get the full resolution and full domain of the data.
 - h. Click **Create Display**. The 21:00 UTC 0.6465 μm image is displayed in the **Main Display** window.
3. Use the zooming and panning controls in the left toolbar to inspect the image. Specifically, notice the fires in Northern California and Southern Oregon.
 - a. Reproject the image with a pre-defined domain. Select **Projections -> Predefined -> US -> CONUS**. Use the scroll wheel to zoom into the image or *Shift+Left Click+Drag* a box around the image.
 - b. Turn off the **Auto-set Projection** option under the **Projections** menu. When this option is checked, the projection will automatically change to the native projection of the new layer. When this option is unchecked, all new layers will be reprojected into the current projection.
 4. Return to the *Field Selector* to load an atmospheric temperature image.
 - a. Select **4.5454 μm Atmospheric Temperature -> Temperature**.
 - b. In the *Advanced*, click the green four arrow button to get the full resolution and full domain of the data.
 - c. Click **Create Display**. The 4.5454 μm temperature image is overlaid on top of the visible image. To see the visible image, turn off the top *HDF MOD02 1km file (All Bands...)* checkbox in the **Legend**.
 5. Use the cursor readout option to see the temperature values for the fire pixels.
 - a. Turn the *HDF MOD02 1km file (All Bands...)* checkbox in the **Legend** back on.
 - b. Click and drag the middle mouse button to activate the **Cursor/Data Readout** option and list the latitude, longitude, and data value under the cursor. Since there are two layers in the main display, there are two different readouts with the appropriate values and units. Notice the high 4.5454 μm temperature values are located at the location of the fire. The data probe from the cursor/data readout option shows that values in the fire are approximately 50 degrees K warmer than outside of the fire.
 6. Edit the maps in the display by using the options in the *Layer Controls*.
 - a. Click on *Default Background Maps* in the **Legend** to go to the *Layer Controls*. The map controls will have two tabs. The first **Maps** tab lists the available maps, and the second **Lat/Lon** tab allows the user to control latitude and longitude lines and labels. At the bottom of both tabs, there is a **Position** slider that allows you to control the vertical positioning of the maps in the **Main Display**.
 - b. In the **Maps** tab you can remove a map, change its visibility, line width, style and color. Use these options to create your own map display.
 - c. If you choose, also add latitude and longitude lines and labels in the **Lat/Lon** tab.

- d. To save a map configuration as the default, in the **Layer Controls** select **File -> Default Maps -> Save as the Default Map Set**. The next time you open a new tab, window, or start McIDAS-V, the defaults will reflect what you selected.
7. Notice that both layers are labeled the same in the **Legend**. Change the text of the labels in the **Main Display**.
 - a. *Right Click* on the top **HDF MOD02 1km files (All Bands...)** label in the **Legend**.
 - b. Choose **Edit -> Properties....** Enter the following for the **Legend Label**:
 - **Legend Label** – 4.5454 um - AQUA

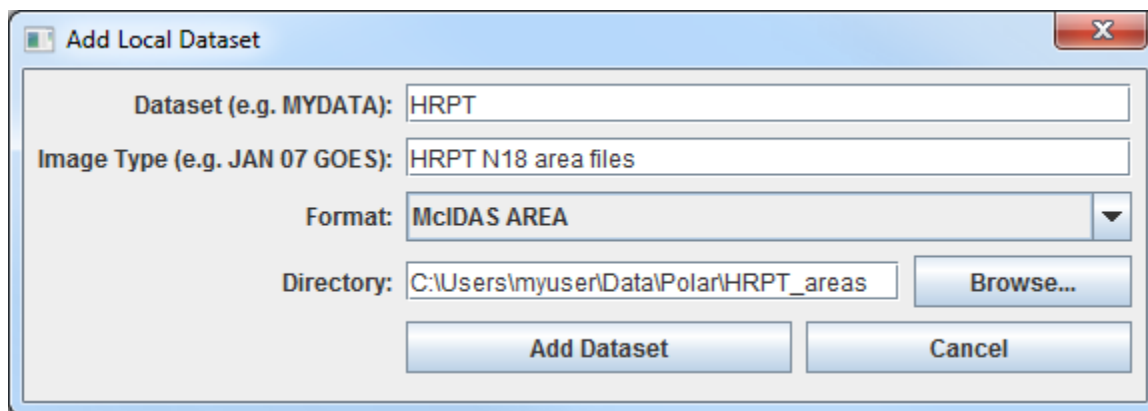


8. Return to the **Data Sources** tab of the **Data Explorer** to load another time.
 - a. In the **Data Sources** tab of the **Data Explorer**, select the **Satellite -> Imagery** chooser with **<LOCAL-DATA>/MODIS**, the **Image Type HDF MOD02 1km files**, and an **Absolute** time of **20:55 UTC**.
 - b. Click **Add Source** to show the **Field Selector**.

- c. A new data source is created in the **Field Selector**. To distinguish it from the first data source, the label is editable. *Right Click* on the highlighted *HDF MOD02 1km files (All Bands...)* and choose **Properties**. In the **Name** field enter, *HDF 20:55 UTC MOD02 1km*. Click **OK**.
- d. Select *0.6465 um Land/Cloud Boundaries -> Brightness*.
- e. In the **Advanced**, click the green four arrow button to get the full resolution and full domain of the data.
- f. Click **Create Display**. The 20:55 UTC 0.6465 μm image is displayed in the **Main Display** window with the other displays also visible (because the **Auto-set Projection** setting is unchecked).
- g. In the **Legend**, click on the trash can next to *4.5454 μm AQUA* to remove the atmospheric temperature display (the middle one listed with the color table range of 180 to 320).

Displaying Local NOAA-18 Data

9. Create a local dataset to access the HRPT (High Resolution Picture Transmission) NOAA-18 Area polar orbiter imagery files on your local machine.
 - a. In the **Main Menu Bar** of the **Main Display** window, select *Tools -> Manage ADDE Datasets*. This will open the **ADDE Data Manager**.
 - b. Select *File -> New Local Dataset*. Enter the following parameters to set up the dataset:
 - **Dataset** – *HRPT*
 - **Image Type** – *HRPT N18 area files*
 - **Format** – *McIDAS AREA*
 - **Directory** – select the *<local path>/Data/Polar/HRPT_areas* directory.



- c. Click **Add Dataset**.
 - d. Close the **ADDE Data Manager** by clicking **Ok** or select *File -> Close*.
10. Remove all layers and data sources and display HRPT 10.8 μm Temperature data.

- a. In the **Main Display** window, select *Edit -> Remove -> All Layers and Data Sources*.
- b. In the **Main Display** select *File -> New Display Tab -> Map Display -> One Panel*.
- c. In the *Satellite -> Imagery* chooser of the **Data Sources** tab of the **Data Explorer**, select *<LOCAL-DATA>* for the **Server**, select *HRPT* for the **Dataset**, and click **Connect**.
- d. Choose the *HRPT N18 area files* **Image Type** and select an **Absolute** time of *04:45:04 UTC*.
- e. Select *Lat/Lon* for the **Navigation**.
- f. Click **Add Source** to show the *Field Selector*.
- g. Select *10.8 um Surface/Cloud-top Temp -> Temperature*.
- h. Under **Displays** click on the *Advanced* tab. Click on the green arrow icon next to **Image Size** to load the full image. Change the **Magnification** to *1 X 1* with the sliders, and click **Create Display**. (The **Magnification** of *1 X 1* downloads the full resolution of the data. Negative magnification values will reduce the amount of data downloaded. For example, *-5 X -5* will sample the data, and every fifth line and every fifth element is sent from the server.)
- i. Change the display to the World projection through the *Projections -> Predefined -> World* menu item. Use *Shift+Left-Click+Drag* to zoom in over the data.

Exercise

Using the *<local path>/Data/Polar/MOD02/** data, create a local dataset, and display an animation of 5 times of data. Disable auto-set projection in the a new display tab and display the data using Line/Element navigation so the data will plot at the correct location for each timestep. These are all daytime images, so displaying any visible or infrared band will produce good images.

Hyper-Multispectral Data

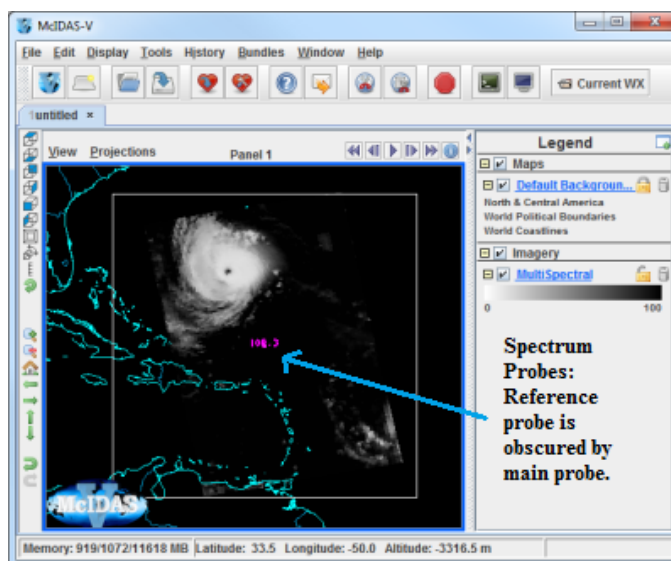
Create a Multispectral Display (AIRS Hyperspectral Data)

1. In the **Main Display** remove all layers and data sources via the *Edit -> Remove -> All Layers and Data Sources* menu item.
2. Display the data from the AIRS file dated September 11, 2018 at 17:29:21 UTC.
 - a. In the **Data Sources** tab of the **Data Explorer**, open the *Satellite -> HYDRA* chooser.
 - b. Navigate to *<local path>/Data/Hydra/AIRS.2018.09.11.175.L1B.AIRS_Rad.v5.0.23.0.R18254140414.hdf*, and click **Add Source**.
 - c. The file loads into the **Field Selector** with the name of Hydra. To make the name more descriptive, edit the properties of the data source.
 - Under **Data Sources**, *Right Click* on *Hydra*.
 - Select **Properties**.
 - Change **Name** from *Hydra* to *AIRS 2018/09/11 17:29:21 UTC*.
 - Click **OK**.
 - d. In the **Field Selector**, under **Fields**, select *MultiSpectral -> Radiance*.
 - e. In the **Field Selector**, under **Displays**, select *MultiSpectral Display*. Click **Create Display**.
3. The MultiSpectral image displays in the **Main Display**. There are four aspects to the **MultiSpectral** display:

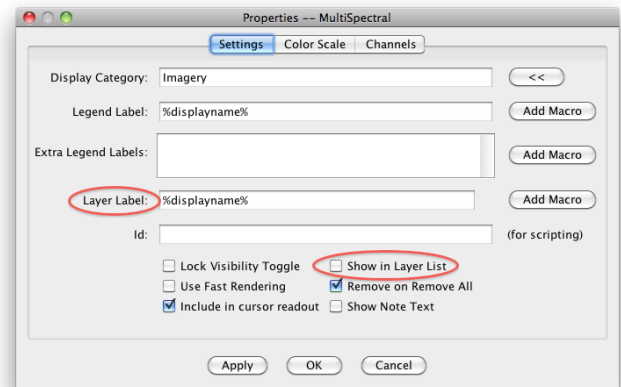
- The image in the **Main Display**.
- The main spectrum probe in the **Main Display** is the magenta square.
- The reference spectrum probe in the **Main Display** is the light blue square. (NOTE: The two spectrum probes may overlap, with only the magenta square visible.)

Left Click+Drag either box to view spectra measured in various pixels of image.)

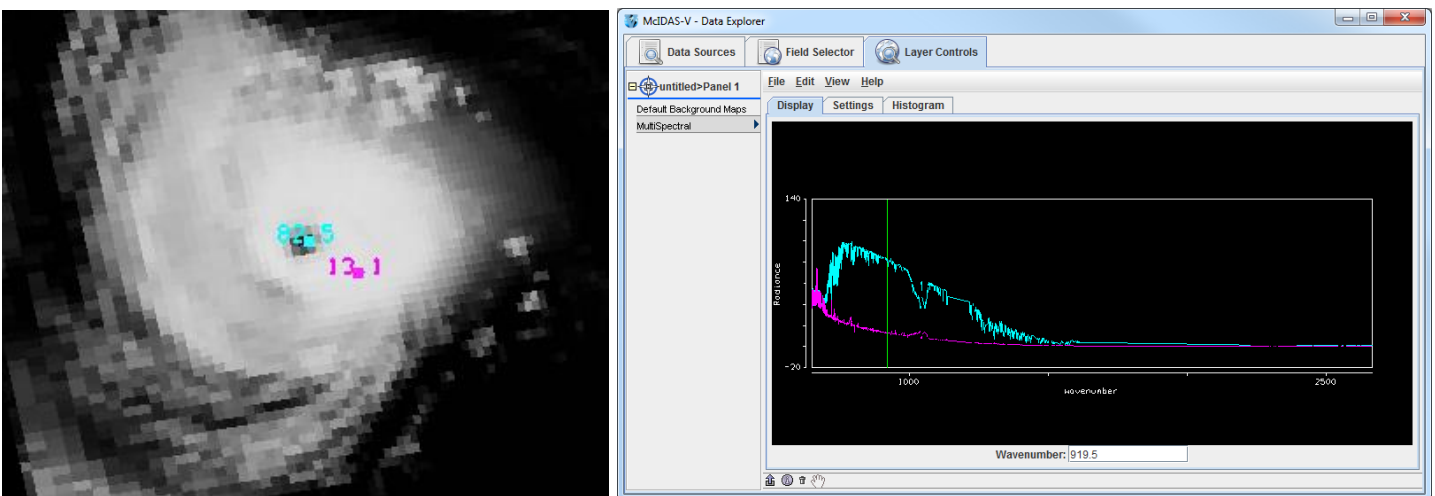
- The spectra of the *MultiSpectral* layer in the **Layer Controls**. The default wavenumber is 919.50 cm^{-1} ($10.8 \mu\text{m}$). From the **Main Display**, click *Multi Spectral* in the **Legend** to view spectra for the AIRS file in the **Layer Controls** tab of the **Data Explorer**.



4. Add an image description to the **Main Display**.
 - a. In the **Layer Controls**, select **Edit -> Properties**.
 - b. In **Layer Label** field, remove `%displayname%`. Type `%datasourcename%`. The macro `%datasourcename%` is linked to the description entered in step 2c.
 - c. If the **Show in Layer List** box is not checked, click to activate this option.
 - d. Click **OK**. The customized label appears at the bottom of the **Main Display**.



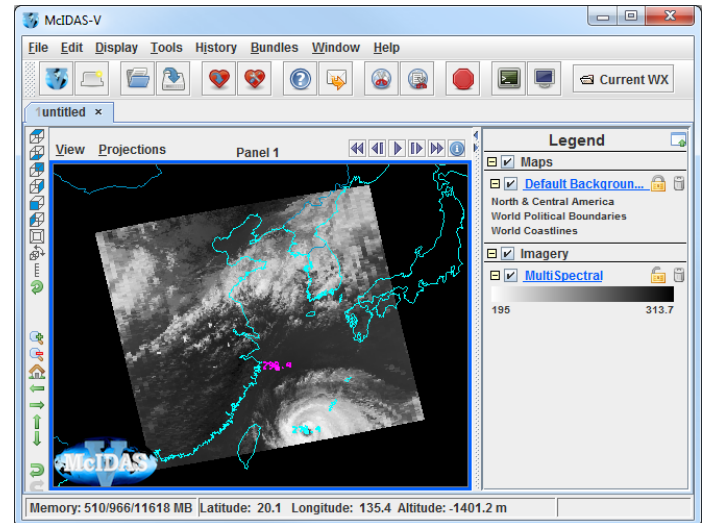
5. Compare clear vs. cloudy spectra by moving the spectrum probes.
 - a. In the **Main Display**, **Shift+Left Click+Drag** to zoom the satellite image.
 - b. Place one probe over a cold cloud top and one over a dark, cloud-free pixel (such as the eye of Hurricane Florence).
 - c. View spectra in the **Layer Controls**. (When in the **Main Display**, click on **MultiSpectral** in the **Legend** to bring **Layer Controls** to the front.)
 - d. Zoom into interesting spectral regions **Shift+Left Click+Drag**. **Control+Left Click** returns the full spectra.
 - e. Choose different wave numbers by **Left Click+Dragging** the green selector bar or by entering a Wavenumber below the spectra.



Create a MultiSpectral Display (MODIS data)

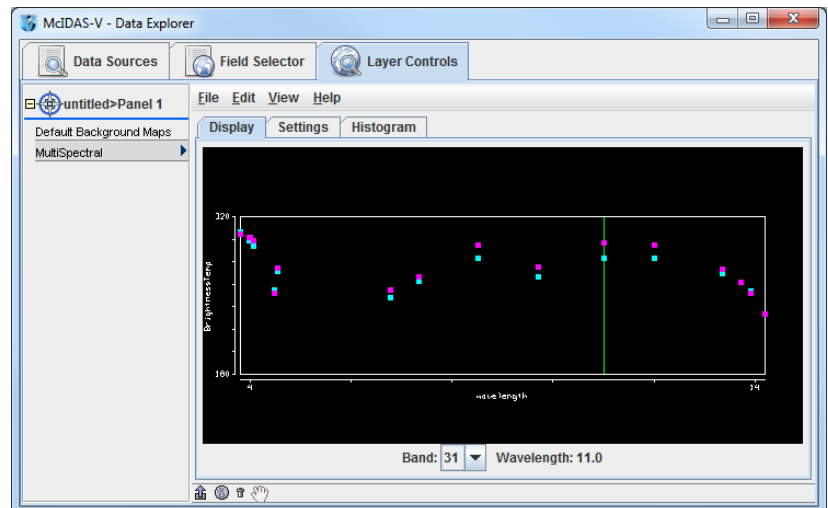
6. Remove all layers and data sources from the previous displays.
7. Display the data from the
<local path>/Data/Hydra/MYD021KM.A2018191.0505.006.2018191160917.hdf file.

- a. In the left panel of the *Data Sources* tab of the **Data Explorer**, choose *Satellite -> HYDRA*, navigate to the MODIS file and click **Add Source**.
- b. Change data source name. In the *Field Selector*, *Right Click* on *Hydra* under **Data Sources** and select **Properties**. Change Name from *Hydra* to *MODIS 2018/07/10 at 05:05:00 UTC* and click **OK**.
- c. Under **Fields** expand the *MultiSpectral* drop down list and select *Emissive_Bands*.
- d. Under **Displays** select *MultiSpectral Display*.
- e. Click **Create Display**.



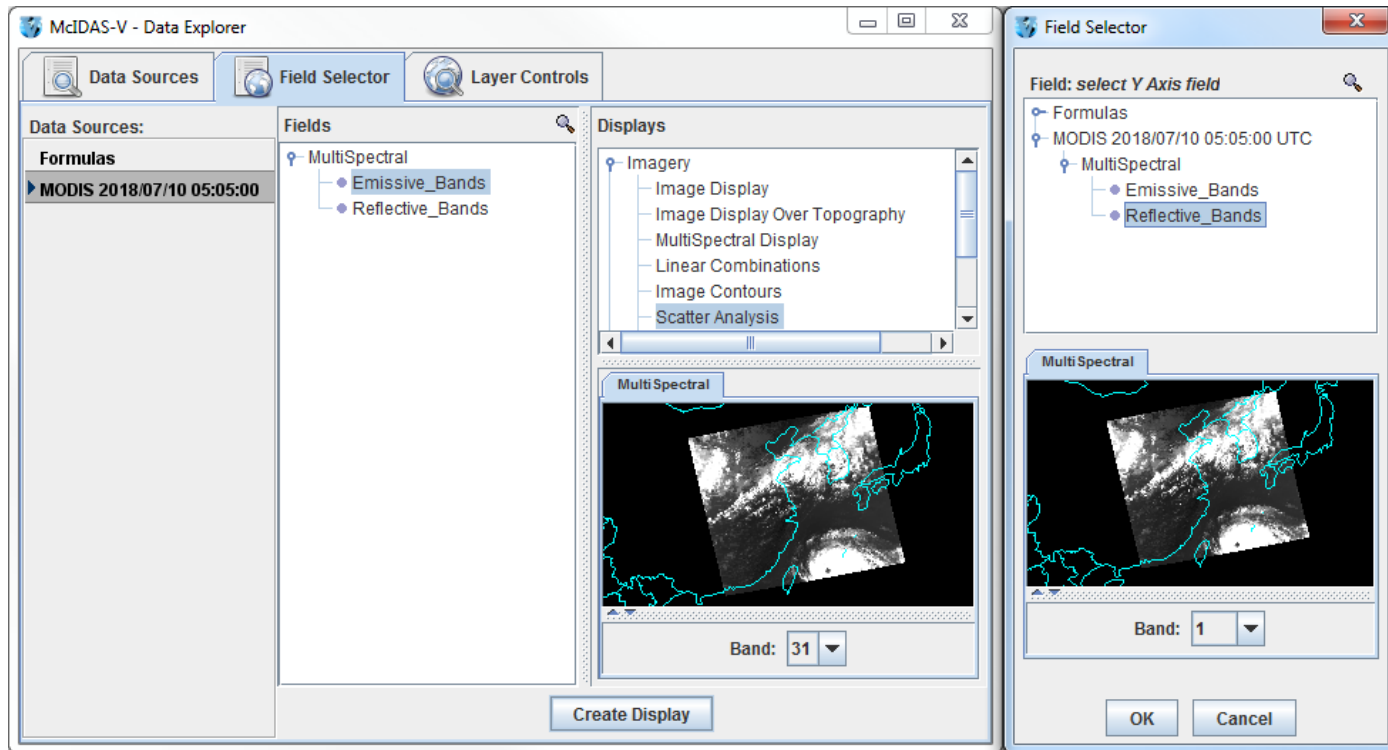
An 11.0 μm image over Eastern China from 05:05:00 UTC on September 07, 2018 is displayed.

8. Change the band displayed in the **Main Display**.
 - a. *Left Click* *MultiSpectral* in **Legend** to bring *Layer Controls* to front.
 - b. Change band with either *Left Click+Drag* option to move the green selector bar or select the **Band** from the drop-down list at the bottom of the *Display* tab of the *Layer Controls*.

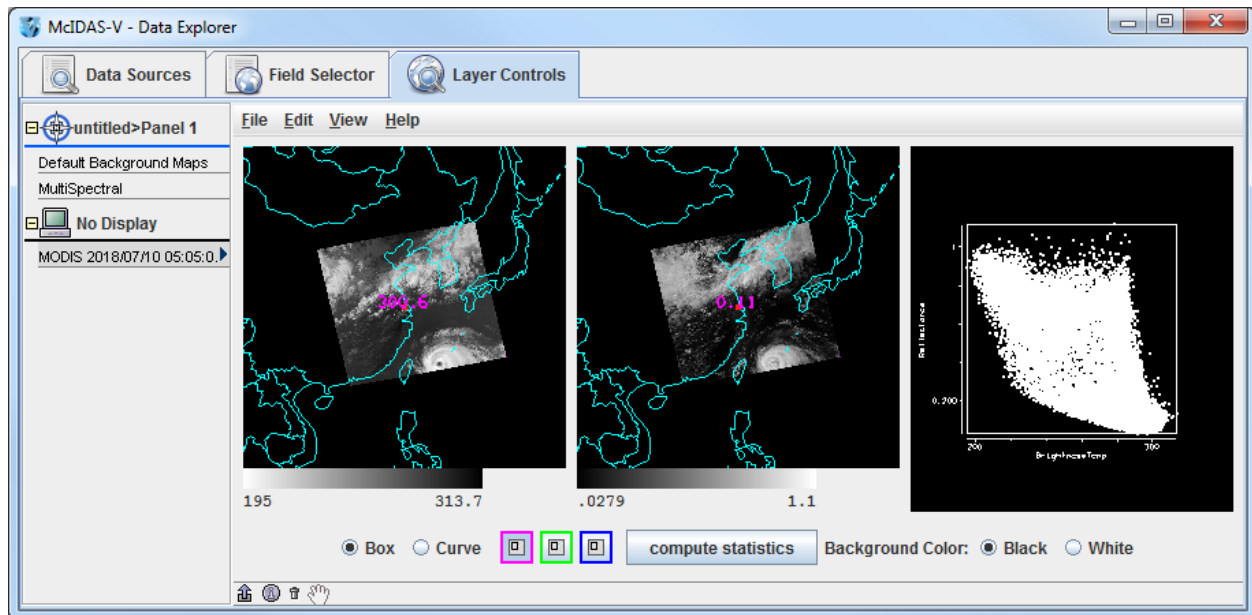


Create a Scatter Analysis of two bands

9. Make a scatter analysis (plot) of two bands.
 - a. In **Data Explorer**, navigate to the **Field Selector** tab. If the **Data Explorer** is not visible, select **Window -> Show Data Explorer** from the **Main Display**.
 - b. Select the **Emissive_Bands** field. This will be the x-axis in this scatter analysis. Under **Displays**, select **Imagery -> Scatter Analysis**.
 - c. Use the **Band** dropdown in the **MultiSpectral** tab in lower right panel to select **Band 31 (11.0 μ m)**. Click **Create Display**.
 - d. A **Field Selector** window will appear for selection of the y-axis data field. In this new **Field Selector**, select **Reflective_Bands**, and then **Band 1 (0.65 μ m)** in the **MultiSpectral** tab. Click **OK**.
10. The scatter analysis will display in the **Layer Controls**.
 - The left image is the first selected field (**Emissive_Bands**).
 - The middle image is the second selected field (**Reflective_Bands**).
 - The right plot is the two-field scatter analysis. The x-axis (abscissa values) corresponds to the first field selected and the y-axis (ordinate values) corresponds to the second field selected. (In this example, band 31 (11 μ m) versus band 1 (0.65 μ m)).



11. Analyze the scatter plot and images by highlighting a box or region in any of three panels.



- To draw a box, click the **Box** radio button and select one of the colors. Use *Shift+Left Click+Drag* option to draw box over a region of the scatter plot. The corresponding points are highlighted in the image.
- Conversely, select a region of interest in one of the images. The corresponding points are highlighted in the scatter plot.
- To draw a shape, click the **Curve** radio button. Use *Left Click+Drag* option to draw an enclosed shape.
- To remove a color from the scatter plot, *Double Click* in that color's selection box.

12. Click the **compute statistics** button at the bottom of the **Layer Controls**. A window with the scatter plot statistics will appear. Also, there is an option to save the statistics as a CSV file.

Exercise

Create a MultiSpectral display of the Brightness Temperature field of the *<local path>/Data/HYDRA/AIRS.2019.09.01.071** file. Set the Wavenumber to display to 1043. Place one probe over the eye of Hurricane Dorian, and one over the hurricane's clouds. Compare the spectral output. Add a new probe and place it over cloud-free land (for example, eastern Cuba).

Radar Data

Displaying Level II radar images: 2D and 3D reflectivity swaths

The **Radar Sweep View in 2D** display creates a 2-dimensional display of 3-dimensional radar data at individual elevation angles. While only data from only one elevation angle can be displayed at once, you can cycle through different elevation angles to help to determine the 3-dimensional characteristics of the data.

The **Radar Sweep View in 3D** display creates a 3-dimensional display of radar data at individual elevation angles. While only data from only one elevation angle can be displayed at once, you can cycle through different elevation angles to inspect the 3-dimensional characteristics of the data. This display appears to be identical to the **Radar Sweep View in 2D** display until the image is rotated, giving a perspective of altitude of the data.

1. From the **Data Explorer**, navigate to the *Radar -> Level II -> Local* chooser.
2. Navigate to the *<local path>/Data/Radar* directory. Use *Shift+Left-Click* to select all of the KMLB* files
3. Click **Add Source**.
4. Open a two panel map display and change the color of the outline around the active panel.
 - a. In the **Main Display**, select *File -> New Display Tab -> Map Display -> Two Panels* menu item.
 - b. By default, the right panel is highlighted by a blue box, which means this is where the data will be displayed. Click the left panel, and you should see the blue box outline the panel display, indicating it is the active panel. Any displays created will be added to the active panel.
 - c. To change the outline color, select *Edit -> Preferences*. Under the *Display Window* tab, there is a *Color Scheme* section on the right side. To the right of *Selected Panel*, click the blue box to bring up the **Set Selected Panel Border Color** window. Select a new color from the swatches and click **OK**. Click **OK** to close the **User Preferences** window. Switch between panels to refresh the box to the selected color.
5. Display the reflectivity data using the **Radar Sweep View in 2D** display type in the left panel.
 - a. Select the left panel of the **Main Display**, and return to the *Field Selector* tab of the **Data Explorer**.
 - b. Under **Fields**, select *Reflectivity*. The available displays will list under **Displays**. Choose the *Radar Displays -> Radar Sweep View in 2D* display type.
 - c. Click **Create Display**.
6. Display the reflectivity data using the **Radar Sweep View in 3D** display type in the right panel.
 - a. Select the right panel of the **Main Display**, and return to the *Field Selector*.
 - b. Under **Fields**, select *Reflectivity*. The available displays will list under **Displays**. Choose the *Radar Displays -> Radar Sweep View in 3D* display type.


- c. Click **Create Display**.
7. There is no visible difference between the two layers until the view is rotated away from the top. *Right Click+Drag* or use the navigation toolbar in the left side of the **Main Display** to rotate the display. Turn looping on and off as you navigate through the 3D reflectivity image.
8. A default for multi-panel displays is shared views, which means that as one panel's navigation changes, they all move in sync. To turn this feature off, go to a panel and uncheck *Projections -> Share Views*.

Displaying realtime Level II radar data

1. From the **Data Explorer**, navigate to the *Radar -> Level II -> Remote* chooser.
2. Using the default catalog <http://thredds.ucar.edu/thredds/radarServer/catalog.xml>, click **Connect**.
3. For **Collection**, select *NEXRAD Level II Radar from IDD*.
4. *Left-Click* on a station currently experiencing precipitation. Note that you can zoom in pan the map with either the mouse or the buttons below the mouse. By default, **Declutter** is enabled, meaning not all stations are shown.
5. In the *Relative* tab of the **Times** panel, make sure that 5 is entered in the **Number of times** field.
6. Click **Add Source**.
7. In the **Main Display** window, create a new 1-paneled map display.
8. Create a 2D display of radar reflectivity.
 - a. In the **Fields** panel of the *Field Selector*, select *Reflectivity*.
 - b. In the **Displays** panel of the *Field Selector*, select *Radar Displays -> Radar Sweep View in 2D*.
 - c. Click **Create Display**.
9. Overlay the 2D data in the **Main Display** with a vertical cross section of reflectivity.
 - a. In the **Fields** panel of the *Field Selector*, select *Reflectivity*.
 - b. In the **Displays** panel of the *Field Selector*, select *Radar Displays -> Radar Cross Section*.
 - c. Click **Create Display**.
 - d. Maneuver the endpoints of the cross section by *Left-Click+Drag* over one of the squares at the ends of the line, or the entire cross section by *Left-Clicking+Dragging* over the triangle at the center of the cross section. Rotate the display with *Right-Click+Drag* or look at the **Layer Controls** to see the 3D data.

Point Observation Data

Plotting and Contouring Surface Point Observations around the world

1. Remove all layers and data sources from the previous displays. If you have created more than one tab, close the extra tabs by clicking on the “X” in the right corner of the tabs.
2. From the *Data Sources* tab of the **Data Explorer**, select the *Point Observations* -> *Plot/Contour* chooser.
 - a. Connect to *adde.ssec.wisc.edu* **Server** with the *RTPTSRC Dataset*.
 - b. Choose the *SFCHOURLY - Real-Time SFC Hourly Point Type*.
 - c. In the **Number of times** text entry box of the *Relative* tab, enter *1* and click **Add Source**.
 - d. In the *Field Selector*, select *Point Data* in the **Fields** panel.
 - e. Select the *Point Data* -> *Point Data Plot* display type.
 - f. In the lower-right panel, click the *Layout Model* tab and click the double down blue arrows next to METAR to select *Observations* -> *Temperature* to plot all available temperatures.
 - g. Click **Create Display**. The default display is a world projection with temperatures plotted from around the world. As you zoom in you will see more stations appear in the plot.
3. Reset the display projection using the Reset Display Projection button  on the left side of the **Main Display** or through the *Projections* -> *From Displays* menu item.
4. Return to the *Data Sources* tab of the **Data Explorer** and select the *Satellite* -> *Imagery* chooser.
 - a. Connect to the *lead.unidata.ucar.edu* **Server** with the *RTGOESR Dataset*.
 - b. Select an **Image Type** of *FDC14 – GOES-16 Full Disk 11.2 um IR Imagery/SST/clouds/rainfall*.
 - c. In the **Number of times** text entry box of the *Relative* tab, enter *1* and click **Add Source**.
 - d. In the *Field Selector* tab, expand the *11.2um* field, and select *Temperature*.
 - e. Select the *Imagery* -> *Image Display* display type. Click **Create Display**.
5. To contour the temperatures plotted on the screen, return to the *Field Selector*.
 - a. Select *SFCHOURLY* in the **Data Sources** panel.
 - b. In the **Fields** panel, select *Gridded Fields* -> *T*.

- c. Select **Plan Views** -> **Contour Plan View** in the list of displays, and click **Create Display**. The contour properties can be changed in the **Layer Controls** tab of the Data Explorer.
- Note that these contours are drawn from surface observations, which are recorded over land, not water. Therefore, contours will look better if you zoom in over a land area in the **Main Display**.

Creating Time Series of Point Observations

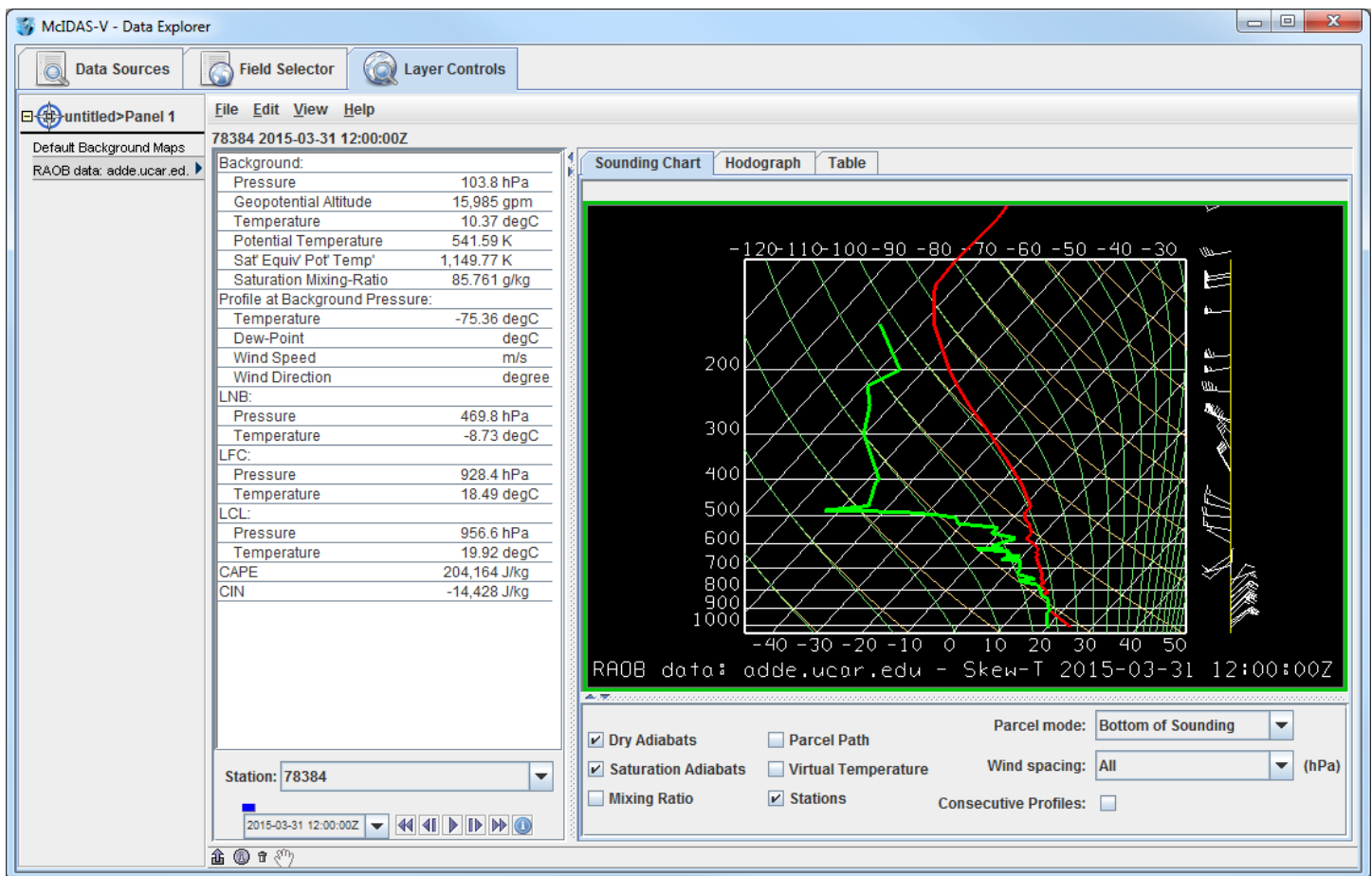
6. Remove all layers and data sources.
7. Return to **Point Observations** -> **Plot/Contour** chooser in the **Data Sources** tab. Use the *adde.ssec.wisc.edu* **Server**, the **RTPTSRC Dataset**, and the **SFCHOURLY Point Type**.
 - a. Select a group of times from the **Absolute** time tab (to select more than one time, use *Shift+Click*), and click **Add Source**.
 - b. Select **Point Data** in the **Fields** panel and click **Create Display** to plot the default station models.
 - c. Zoom in on the display and play through the loop to see how the observations change.
8. In the **Layer Controls**, follow these steps:
 - a. Click on the **Plot** tab to create a time series.
 - b. Zoom in over a station in the **Main Display** and *Left Click* on the center of it. The station information should fill into the table below the plot in the **Layer Controls**.
 - c. Add the temperature to the plot by *Right Clicking* on the **T** field in the table and selecting **Add To Chart**.

Displaying RAOB Sounding Data

9. Remove all layers and data sources.
10. Select the **Point Observations** -> **Soundings** -> **Remote** chooser in the **Data Sources** tab.
 - a. Connect to the *adde.ssec.wisc.edu* **Server** with the **RTPTSRC Dataset**.
 - b. Select **UPPERMAND** for the Mandatory Levels and **UPPERSIG** for the Optional Significant Levels.
11. Select a station for your sounding display.
 - a. Use the zooming and panning buttons to find your station(s).
 - b. Select an available time and click on your station to see if a sounding is available. Available soundings will show up in the **Selected** box on the bottom right of the chooser.

- c. If more than one is available for your station, select the most recent time and add the source.
- d. Under the **Displays** panel of the *Field Selector*, you have the option to choose which type of thermodynamic diagram to display (Skew-T, Stüve, Emagram).
- e. Select one of the thermodynamic diagrams and click **Create Display**.

12. The sounding will show up in a 2D display in the *Layer Controls* tab along with a list of thermodynamic parameters. As you move the mouse over the sounding, the thermodynamic parameters will update.



13. The temperature and dewpoint traces can be modified via the cursor.

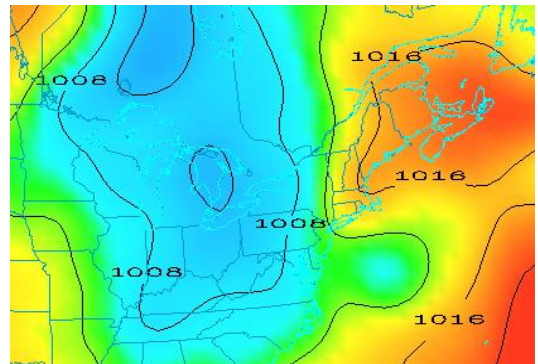
- a. *Left Click* on the first data-point to be modified and drag it horizontally along an isobar to the desired position, then move the pointer diagonally along an isotherm to keep the data-point at the desired position and to pick-up the next data-point.
- b. Repeat until done and then release the mouse button.
- c. To reset the sounding, use the *Edit -> Reset Sounding* menu item of the *Layer Controls* tab.

- d. Use the checkboxes below the sounding to configure the sounding chart. When the **Stations** box is checked, you will see a square box in the **Main Display** indicating the location of each sounding selected in the **Data Sources** tab. Select a new station by *Left Clicking* another square box in the **Main Display**, or by choosing a station in the **Station** dropdown menu in the **Layer Controls**.
- e. In the **Layer Controls**, in addition to the **Sounding Chart**, there are also tabs for a **Hodograph**, and a **Table** of all of the data in the sounding.

Gridded Data


Displaying Gridded data: 2D

1. Remove all layers and data sources from the previous displays.
2. If you are using real-time data, skip to step 3. Otherwise, load the file `<local path>/Data/Grid/GFS_CONUS_80km_20190903_1200.grib1`.
 - a. In the **Data Explorer**, navigate to the *Gridded Data -> Local* chooser.
 - b. Select the `<local path>/Data/Grid/GFS_CONUS_80km_20190903_1200.grib1` file.
 - c. Click **Add Source**.
3. If you are using realtime data, follow the instructions below.
 - a. Select the *Gridded Data -> Remote* chooser from the **Data Sources** tab of the **Data Explorer**.
 - b. From the **Catalog** pull down menu, select <http://www.ssec.wisc.edu/mcidas/software/v/threddsRTModels.xml>.
 - c. Add the *Realtime Data -> NCEP Model Data -> Global Forecast System (GFS) -> GFS-CONUS 80km -> Latest* source.
4. Display the MSL Pressure.
 - a. In the **Field Selector**, select the *2D grid -> Pressure reduced to MSL @ Mean sea level* field.
 - b. Select the **Plan Views -> Color-Shaded Plan View** Display.
 - c. In the **Times** tab, use the dropdown to choose **Use Selected**, and select the first six times.
 - d. Click **Create Display**.
5. In the **Layer Controls**, turn on the option to **Shade Colors**.
6. Go back to the **Field Selector**, change the display type to **Contour Plan View**, and overlay the same field.
 - a. Select the *2D grid -> Pressure reduced to MSL @ msl* field.
 - b. Select the **Plan Views -> Contour Plan View** display type.
 - c. Click **Create Display**. The contours will now overlay the *Color-Shaded Plan View* display.



7. In the **Layer Controls**, click on the **Color Table** name (PressureMSL) to change the color table of the contours to **System -> Solid Colors -> Black**.
8. In the **Layer Controls** tab, click the **Change** button for **Contour**. For **Labels**, change the **Align** item to **Horizontal** and move the **Frequency** slider up to a value of **Hi**. Click **OK**.
9. Zoom in over a region to see a display like that on the previous page.

Displaying Gridded data: 3D

10. Remove all layers. In the **Field Selector**, select the **3D grid -> Temperature @ Isobaric surface** field.
 - a. Select the **Plan Views -> Color-Filled Contour Plan View** display type.
 - b. In the **Times** tab select the first 6 images.
 - c. In the **Level** tab, select **500 hPa**.
 - d. Click **Create Display**.
11. Use the rotating and zooming controls to rotate the **Main Display** window. You can see that the 500 hPa surface is displayed above the map layer.
12. Keep the **Main Display** rotated so the difference in height between the map and the display can be seen.
13. Change the level of the display. Note that when the level is changed, both the plotted values and the z-level of the display change in the **Main Display**.
 - a. Click on **Latest NCEP GFS CONUS...** in the **Legend** to go to the **Layer Controls** for this item.
 - b. Change the level to **850 hPa** by using the **Levels** menu.
 - c. Change the contour interval to 2 Celsius by clicking on the **Change** button in the Contour section of the **Layer Controls**.
14. Reset the projection using the  button on the left side of the **Main Display** window
15. Change the level to **250 hPa** in the **Layer Controls** using the method in 13b. You should notice that the map does not show up on the displayed data.
16. Move the map above the 250 hPa contours and change the map colors to white.
 - a. In the **Layer Controls**, select **Default Background Maps** from the list of layers on the side of the tab.

- b. At the bottom of the *Maps* tab, move the **Position** slider to the z-level of 0.30 to move higher into the 3D display. You should now see that the map lines are drawn on the data displayed in the **Main Display**. However, the map lines may be hard to see since they are the same color as the data by default.
- c. In the *Maps* tab of the *Layer Controls* for Default Background Maps, change the map colors of “North & Central America”, “World Political Boundaries”, and “World Coastlines” to white. Do this by clicking on the colored square next to each of the map lines, selecting white, and hitting **OK**. The map lines should now be more clearly visible in the **Main Display** window.

Displaying Gridded data: Ensembles

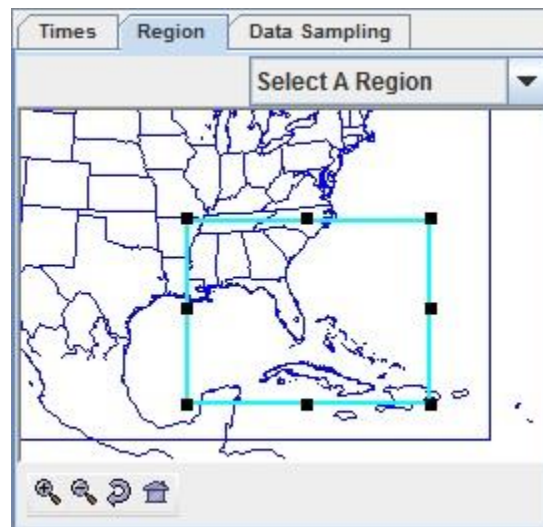
17. Remove all layers and data sources.
18. If you are using real-time data, skip to step 19. Otherwise, load the local NetCDF file `<local path>/Data/Grid/GEFS_Global_1p0deg_members`.
 - a. In the *Data Sources* tab of the **Data Explorer**, navigate to the *Gridded Data -> Local* chooser.
 - b. Select the `<local path>/Data/Grid/GEFS_Global_1p0deg_members.nc`.
 - c. Click **Add Source** and continue to step 20.
19. Add a gridded data source of ensemble members.
 - a. In the *Data Sources* tab of the **Data Explorer**, navigate to the *Gridded Data -> Remote* chooser.
 - b. Select *Realtime Data -> NCEP Model Data -> Global Ensemble Forecasting System (GEFS) -> GEFS Members - Forecasts -> latest** and click **Add Source**.
20. To ensure that the ensemble’s display uses the native projection of the data, verify that *Projections -> Auto-set Projection* is enabled in the **Main Display**.
21. Display MSLP data of a few ensemble members.
 - a. In the **Fields** panel of the *Field Selector*, select the *2D grid -> Mass -> Pressure reduced to MSL @ Mean sea level* field.
 - b. In the *Ensemble* subset tab, select (*command left-click*) members *.0, 5.0, 10.0, 15.0, and 20.0*.
 - c. In the *Times* subset tab, choose **Use Selected**. Then, select the first 10 times.

- d. Click **Create Display**.
22. Inspect the display. Play through the loop to see how the ensemble members slowly begin diverging from each other you get further away from the initialization of the model run.
 23. Note that by default the ensemble contours are colored by their member value (per the enhancement values of the colorbar in the **Legend** ranging from 0 to 20). Change the display to color the contours by pressure value instead of ensemble member.
 - a. In the *Layer Controls* for the ensemble layer, uncheck **Color by Member**.
 - b. Look at the display to notice that all of the contours are now the same color. This is because the range from the ensemble members (0-20) is still being applied to the enhancement. Since all of the pressure values are greater than 20 Pascals, the color value for the enhancement table maximum is used. Change the enhancement values to fit the range of data by *right-clicking* on the colorbar in the **Legend** and selecting *Change Range*. In the **Change Range** window, enter a **From** value of 95200 and a **To** value of 105200.
 24. Use a formula to display the minimum pressure value of a few ensemble members.
 - a. Remove all layers.
 - b. In the *Field Selector* tab of the **Data Explorer**, click on **Formulas**.
 - c. In the **Fields** panel, select the *Grids -> Ensembles -> Ensemble grid lowest values* formula, the *Imagery -> Image Display* display type, and click **Create Display**.
 - d. In the **Field Selector** window, select the *2D grid -> Mass -> Pressure reduced to MSL @ Mean sea level*. In the *Times* tab, select the first 10 times. In the *Ensemble* tab, select members .0, 5.0, 10.0, 15.0, and 20.0. Click **OK**.
 - This generates a display of the first 10 timesteps of the ensemble. The ensemble members selected when evaluating the formula are used to determine the lowest pressure value at each location over each timestep.
 25. The formula in step 24 provides the option for individually selecting which ensemble members are used in the formulas. As an alternative, a data source can be added that already has a derived mean and standard deviation for all ensemble members. Remove all layers and data sources.
 26. If you are using real-time data, skip to step 27. Otherwise, load the local NetCDF file
`<local path>/Data/Grid/GEFS_Global_1p0deg_products.`

- a. In the **Data Sources** tab of the **Data Explorer**, navigate to the **Gridded Data -> Local** chooser.
 - b. Select the `<local path>/Data/Grid/GEFS_Global_1p0deg_products.nc`.
 - c. Click **Add Source** and continue to step 28.
27. In the **Data Sources** tab of the **Data Explorer**, navigate to the **Gridded Data -> Remote** chooser.
- a. In the **Data Sources** tab of the **Data Explorer**, navigate to the **Gridded Data -> Remote** chooser.
 - b. Select **Realtime Data -> NCEP Model Data -> Global Ensemble Forecasting System (GEFS) -> GEFS Derived Forecast Products -> latest*** and click **Add Source**.
28. In the **Field Selector**, select the **2D grid -> Mass -> Pressure reduced to MSL (Unweighted mean of all members) @ Mean sea level**. Note that this tree will not exist if working with the local NetCDF file, only the **Pressure reduced to MSL (Unweighted mean of all members) @ Mean sea level** field.
29. Choose the **Plan Views -> Contour Plan View** display type, the first 10 times, and click **Create Display**.

3D Isosurface Display of Winds over Hurricane Dorian

30. Remove all layers and data sources, create a new tab, and close any previously-existing tabs.
31. Load the `<local path>/Data/Grid/GFS_CONUS_80km_20190903_1200.grib1` file.
- a. In the **Data Explorer**, navigate to the **General -> Files/Directories** chooser.
 - b. Select the `<local path>/Data/Gridded/GFS_CONUS_80km_20190903_1200.grib1` file.
 - c. Click **Add Source**.
32. Display flow vectors over the location of Hurricane Dorian.
- a. In the **Fields** panel of the **Field Selector**, choose the **2D grid -> Derived -> Flow Vectors** field.
 - b. In the **Displays** panel, choose **Flow Displays -> Vector Plan View**.
 - c. In the **Times** tab, select the first 10 times.
 - d. In the **Region** tab, change the dropdown to **Select A Region** and draw a box over the domain pictured here (Florida and out into the Caribbean). Click **Create Display**.



33. Add an isosurface of wind speed to the display.
 - a. In the **Fields** panel of the *Field Selector*, choose the *3D grid -> Derived -> Speed* field.
 - b. In the **Displays** panel, choose *3D Surface -> Isosurface*.
 - c. The *Times* and *Region* tabs should remember the settings specified for the flow vectors in step 32. In the *Level* tab, select *All Levels*.
 - d. Click **Create Display**.

34. Inspect the display and change some properties of the isosurface.
 - a. Use the mouse and navigation toolbars to rotate the display, and the Time Animation Widget to play through the loop.
 - b. By default, the wind speed isosurface is set to 30.7 m/s. Change these units to miles per hour by *Right-Clicking* on the blue text of the 3D surface layer and selecting *Edit -> Change Display Unit*. In the **Change Unit** window, change the dropdown to *mi/hr* and click **OK**. The isosurface is now set to 68.6 miles per hour.
 - c. In the *Layer Controls*, change the **Isosurface Value** to *100* and click Enter to see where the 100mph winds are forecasted to be located.
 - d. In the *Layer Controls*, change the **Isosurface Value** to *50* and click Enter to see where the 50mph winds are forecasted to be located.

Local Files

Plotting Point Observations from a text file

1. Remove all layers and data sources from the previous displays.
2. If there is more than one tab, close the extra tabs by clicking the “X” in the right corner of the tabs.
3. Add the data from the *<local path>/Data/Generic/Point/text/N18_N_2009_090_12_03.asci* file.
 - a. In the *Data Sources* tab of the **Data Explorer**, open the *General -> Files/Directories* chooser.
 - b. In the **Data Type** field, choose **Text Point Data files**.
 - c. Navigate to the *<local path>/Data/Generic/Point/text/N18_N_2009_090_12_03.asci* file. Click **Add Source**.
4. Define the field names and units.
 - a. In the **Point Data** window, select **Space** for the **Delimiter**.
 - b. Under **Start line**, click the down arrow once to skip over the header line.
 - c. In the lower panel of the **Point Data** window, fill in the fields exactly as shown below. Note that the items in these fields are case-sensitive. Do not use the pull down menus, as the choices are very limited.

Point Data

Delimiter: Comma Semicolon Tab Space

Skip Pattern:

Start line:

⏏ type sat day/hms lat lon pre spd dir rff qi



⏏ IR NOAA-18 200903311203 62.35 143.21 700 5.4 241 81.63 0.61

IR NOAA-18 200903311203 62.64 140.49 756 8.4 265 72.72 0.75

Enter the field names and units. Leave name field blank to skip the field Preferences


Value	Name	Unit/Date Format	Missing Value	Extra (e.g., colspan)
IR				
NOAA-18				
200903311203	Time	yyyyMMddHHmm		
62.35	Latitude	degrees		
143.21	Longitude	degrees		
700	Pressure	hPa		
5.4	Speed	m/s		
241	Direction	degrees		
81.63				
0.61				

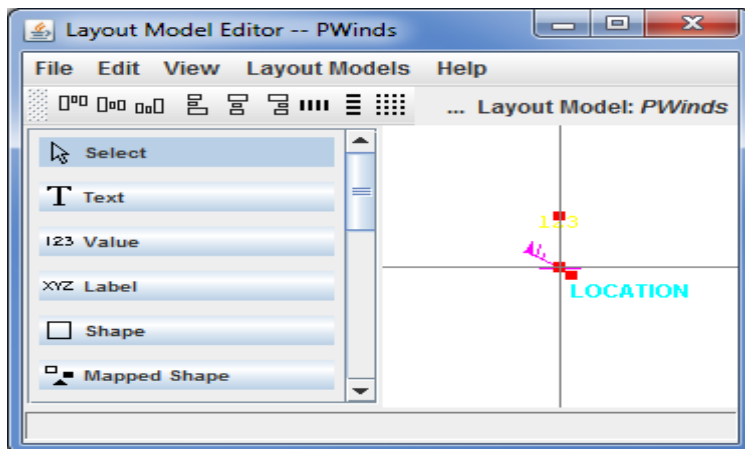
OK Cancel

- d. Select **Preferences** -> **Save Current**. Enter **PWinds** for the **Name** and click **OK**.
 - e. Click **OK** at the bottom of the **Point Data** window.
5. Open the **Layout Model Editor** to define how the data is displayed.
 - a. In the **Main Display** select the **Tools** -> **Layout Model Editor** menu item.
 - b. In the **Layout Model Editor**, select the **Layout Models** -> **Location** menu item to act as a starting point for the new layout model.
 6. Add a windbarb to your layout model.
 - a. In the left panel of the **Layout Model Editor**, click **Windbarb**. *Left Click* in the window to place the windbarb object. *Left Click+Drag* to move the windbarb to the center of the box.
 - b. In the **Display** tab of the **Properties Dialog – WindBarb Symbol** window, click the down arrow  for **U or windspeed parameter**. Select **Current Fields** -> **N18_N_2009_090_12_03.asci** -> **Speed**.
 - c. Click the down arrow  for **V or direction parameter**. Select **Current Fields** -> **N18_N_2009_090_12_03.asci** -> **True Heading Angle – Direction**. Click **OK** to

close the **Properties Dialog** window.

7. Add the pressure value to your layout model.


- a. In the left panel of the **Layout Model Editor**, click **123 Value**. *Left Click* in the window to place the value object. Drag the object to above the windbarb.
- b. In the **Display** tab of the **Properties Dialog – Value Symbol** window, click the down arrow  for **Parameter**. Select **Current Fields -> N18_N_2009_090_12_03.asci -> Pressure -- Pressure**.
- c. Under **Unit**, select **hectoPascals**
- d. Click on **Foreground Color** box and select a yellow color. Click **OK**.
- e. Click **OK** to close the **Properties Dialog** window.



8. Save your new layout model.

- a. In the **Layout Model Editor**, select **File -> Save As**. Enter **PWinds** for the **Layout Model name**. Click **OK**.
- b. Close the **Layout Model Editor** window.

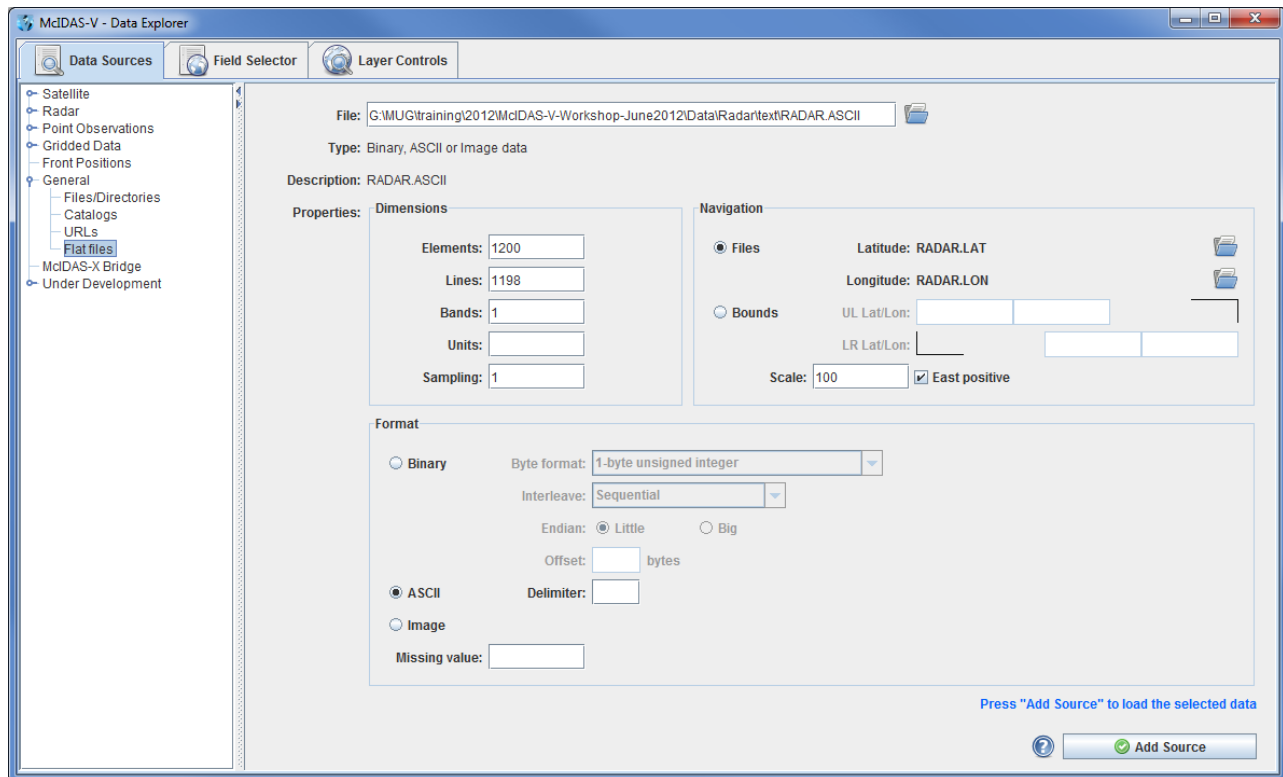
9. Create the display.


- a. In the **Fields** panel of the **Field Selector**, select the **Point Data** field.
- b. Select the **Point Data Plot** display type.
- c. In the lower-right panel of the **Field Selector**, open the **Layout Model** tab. Click the down arrows  and select **PWinds <local>**. Click **Create Display**. The default display is a world projection with wind barbs and pressures plotted. As you zoom in you will see more stations appear in the plot.

Displaying radar imagery from a text file

10. Remove all layers and data sources from the previous displays. If you have created more than one tab, close the extra tabs by clicking the “X” in the right corner of the tab.
11. To add the data source for the `<local path>/Data/Generic/Radar/text/RADAR.ASCII` file, in the **Data Sources** tab of the **Data Explorer**, open the **General -> Flat files** chooser. In the Flat files chooser:
 - Click **Open File** icon next to **File** field and navigate to `<local path>/Data/Generic/Radar/text/RADAR.ASCII`.

- In the **Dimensions** section of the **Properties** panel, enter *1200* for **Elements** and *1198* for **Lines**.
- In the **Navigation** section of this tab, ensure **Files** is selected.
- Click the **Open File** icon for **Latitude** and navigate to *<local path>/Data/Generic/Radar/text/RADAR.LAT*.
- Click the **Open File** icon for **Longitude** and navigate to *<local path>/Data/Generic/Radar/text/RADAR.LON*.
- Change the **Scale** value to *100* and check the **East positive** box.
- In the **Format** section, select **ASCII**.
- Click **Add Source**.



12. Display the radar data.
 - a. In the **Fields** panel of the *Field Selector* tab, click key icon  next to **RADAR.ASCII** to expand the field and select *Flat data*.
 - b. In the **Displays** panel, select *Image Display*.
 - c. Click **Create Display**.
13. The default display is a world projection, but this data is only over Oklahoma in the south-central United States. Change the projection of the **Main Display** to Oklahoma.
 - a. In the **Main Display**, navigate to the *Projections -> Predefined -> US -> States -> N-Z -> Oklahoma* menu item.
 - b. Zoom in on the data by using *Shift+Left Click+Drag* to create a bounding box around the data.
14. Change the color bar to display the radar data using typical colors used for a base reflectivity display.
 - a. In the **Legend**, *Right Click* on the gray scale color bar.
 - b. Select *Radar -> Base Reflectivity 16 Levels*.

Time Matching

Introduction

Time matching is a feature designed to create loops that accommodate data with differing temporal frequencies. Time matching involves setting one layer (generally the layer with the lower temporal frequency) as the time driver, and other layers can be set to match the times of the time driver. An example use case for this would be creating a loop of five hours of METAR surface observations at hourly intervals, and then overlaying each METAR with a satellite image that matches up closest with the METAR times. Generally, satellite data is available in time intervals of less than an hour, so instead of having to manually select each satellite image that matches the METAR times, McIDAS-V will do this automatically. This tutorial will walk through this example.

There are two locations to set a layer to be the time driver:

- In the **Times** tab of the **Field Selector** use the dropdown to select **Set As Time Driver**, as demonstrated in this tutorial.
- In the **Layer Controls** tab of the **Data Explorer**, use the **View -> Times -> Set As Time Driver** menu item.

A time driver can also be set through the Time Animation Widget, by setting a block of user-defined times to be the time driver.

Dependent on the chooser, there may be three locations of setting a layer to match the time driver, assuming a time driver has been set:

- In the **Data Sources** tab of the **Data Explorer**, select the **Match Time Driver** checkbox, as demonstrated in this tutorial. This option is available through the **Satellite -> Imagery**, **Radar -> Level II -> Remove**, **Radar -> Level III**, and **Point Observations -> Plot/Contour** choosers.
- In the **Times** tab of the **Field Selector** use the dropdown to select **Match Time Driver**.
- In the **Layer Controls** tab of the **Data Explorer** use the **View -> Times -> Match Time Driver**. Note that if using this option, the layer must be reloaded with **File -> Reload Data**.

Displaying Surface Observation Data and Matching Satellite Times

1. In the **Main Display**, remove all layers and data sources via the **Edit -> Remove -> All Layers and Data Sources** menu item.
2. Create a new tab with the **File -> New Display Tab -> Map Display -> One Tab** menu item.
3. Load in the five most recent surface hourly observations.

- a. In the *Data Sources* tab of the **Data Explorer**, navigate to the *Point Observations -> Plot/Contour* chooser.
 - b. For **Server**, select *adde.ssec.wisc.edu*, and for **Dataset** select *RTPTSRC*.
 - c. Click **Connect**.
 - d. For **Point Type**, select *SFCHOURLY – Real-Time SFC Hourly*.
 - e. In the **Times** panel, select the *Relative* tab. For **Number of times**, enter 5 and click **Add source** to create a data source of the 5 most recent times.
 - f. Click **Add Source**.
4. Display METARS, and set the layer to be the time driver.
 - a. In the **Fields** panel of the *Field Selector*, select the *Point Data* field.
 - b. In the **Displays** panel, choose the *Point Data -> Point Data Plot* display type.
 - c. In the *Times* tab, use the dropdown menu to select *Set As Time Driver*.
 - d. In the *Layout Model* tab, select the *Observations -> METAR* layout model.
 - e. Click **Create Display**.
5. Load in GOES-16 11.2um IR satellite images that match the METAR times.
 - a. In the *Data sources* tab of the **Data Explorer**, navigate to the *Satellite -> Imagery* chooser.
 - b. For **Server**, select *lead.unidata.ucar.edu*, and for **Dataset** select *RTGOESR*.
 - c. Click **Connect**.
 - d. For *Image Type*, select *CONUSC14 – GOES-16 CONUS 11.2 um IR Imagery/SST/clouds/rainfall*.
 - e. In the **Times** panel, at the bottom of the *Relative* tab, select the **Match Time Driver** checkbox.
 - f. Click **Add Source**.
6. Display the Temperature field of the satellite data.

- a. In the **Fields** panel of the *Field Selector*, select the *11.2 um IR Imagery, SST, clouds, rainfall -> Temperature* field.
- b. In the **Displays** panel, select *Imagery -> Image Display*.
- c. Click **Create Display**.

Note: The METAR data is in hourly time steps. The satellite data on the server, when in the default scanning mode, comes in every 5 minutes (can be seen in the *Satellite -> Imagery* chooser by selecting the *Absolute* times tab). The **Match Time Driver** checkbox in the *Satellite -> Imagery* sorted through the list of times, to only load in the time steps that matched closest to the hourly METAR data. This allowed for the creation of a display where both the METAR and satellite data were displayed in hourly intervals.

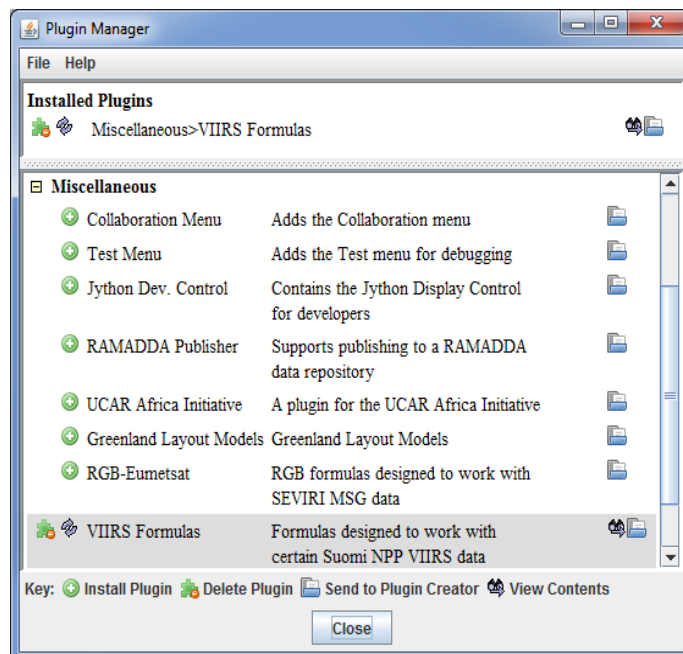
Under Development - JPSS

Introduction

McIDAS-V has the ability of displaying JPSS (Suomi NPP and NOAA-20) data through the *Under Development -> Imagery – JPSS* chooser. Among the JPSS products that McIDAS-V can display are data from the VIIRS (all bands of SVM and SVI, EDRs, Day/Night band), CrIS, and ATMS instruments. In order for the data to be recognized by McIDAS-V, the geolocation files must be contained within the same directory as the data. Some sources of data, including NOAA's CLASS, package the data and geolocation files together by default. Other sources of JPSS data may store the geolocation in a separate file from the data. Note that this chooser is still under development and improvements in functionality as well as the ability to utilize different JPSS products will be added in the future.

Installing the VIIRS Formulas Plugin

1. This tutorial uses two formulas specifically designed to work with Suomi NPP VIIRS data. These formulas are not packaged as part of the core group of formulas in McIDAS-V, and must be added via a plugin. Add this plugin.
 - a. In the **Main Display** window, navigate to the *Tools -> Plugins -> Manage* menu item to open the **Plugin Manager**.
 - b. The top of the **Plugin Manager** lists plugins currently installed, and the bottom panel lists out various plugins available for installation. Expand the *Miscellaneous* tree and click the green **Install Plugin** button next to *VIIRS Formulas*.
 - c. This pops up a message informing that McIDAS-V must be restarted to complete the installation of the plugin. Click **OK** through this message and restart McIDAS-V.



Display a Single Granule of NOAA-20 Data

2. Load a single granule of NOAA-20 SVM14 VIIRS data.
 - a. In the *Data Sources* tab of the **Data Explorer**, select *Under Development -> Imagery – JPSS*.
 - b. Under **Files**, select the following file:

*<local path>/Data/JPSS/SVM14/SVM14_j01_d20190828_t1716290**

Note that this file contains just the data (SVM). The matching terrain-corrected-geolocation file (GMTCO) is also in this directory. On some platforms, these GMTCO files might not be visible in the JPSS chooser. On other platforms they may be listed, but grayed-out. McIDAS-V can also work with JPSS files that package the data and geolocation in the same file, such as those distributed by default from NOAA CLASS.

- c. Click **Add Source**.

Note that the first time a JPSS granule is added through this chooser, a **Plugin Compatibility Notice** window may appear letting users of the older version of the VIIRS Formulas plugin know that a new one is available. In step 1 above the most recent version of this plugin was installed. Click the *Do not show this message again* checkbox and click **OK**.

3. Display Brightness Temperature data at low resolution.
 - a. In the *Field Selector* tab of the **Data Explorer**, select the **VIIRS-M14-SDR_All/BrightnessTemperature** field, and the **Imagery -> Image Display** display type.
 - b. Note that the **Region** tab in the *Field Selector* shows a preview image of the selected field.
 - c. Click **Create Display**.
 - d. Investigate the display and note that the resolution of the display is coarse, not the native 750 meter resolution of the data. This is because no subsetting of the data was done in the **Region** tab of the *Field Selector*.
4. Display Brightness Temperature data at the full resolution of the data.
 - a. From the **Region** tab of the *Field Selector*, use *Shift+Left-Click+Drag* to draw a green box in the data. Any data contained within this box will be displayed at full resolution. Note that the entire box must be contained within the granule's data.
 - b. Click **Create Display**.
 - c. Investigate the display and note that the resolution of the data has improved. However, note that the bowtie deletion lines still exist on the east and west edges of the granule.
5. Use the swathToGrid formula to display the Radiance data at full resolution with the bowtie deletion lines removed.

- a. In the *Field Selector* tab of the **Data Explorer** select **Formulas**.
- b. Choose the *swathToGrid* formula, the *Imagery -> Imagery Display* display type, and click **Create Display**. Note that this formula was added as part of the plugin installed in step 1 above.
- c. In the **Select input** window, enter a value of 750 for **resolution in meters** and click **OK**.
- d. In the **Field Selector** window, select the *VIIRS* -> Image -> VIIRS-M14-SDR_All/Radiance* field.
- e. Wait for the *Region* tab to appear at the bottom of the **Field Selector** window showing a preview image of the granule and click **OK**.
- f. Investigate the display and notice that the data is now displayed at full resolution with the bowtie deletion lines removed. Make the clouds white by *Right-Clicking* on the enhancement in the **Legend** and selecting *System -> Inverse Gray Scale*.
- g. When you are done investigating the display, remove all layers and data sources.

Create a RGB Display with Aggregated Granules

McIDAS-V has the ability to aggregate several small granules into one large one by selecting multiple time-consecutive granules in the *Data Sources* tab of the **Data Explorer** before clicking **Add Source**. Multiple bands can also be included in the same data source if the times match up between the bands.

6. Create a data source of three time-consecutive granules of three visible SVM bands.
 - a. In the *Data Sources* tab of the **Data Explorer**, select *Under Development -> Imagery – JPSS*.
 - b. Navigate to the *<local path>/Data/JPSS/RGB* and use *Shift+Left-Click* to select all of the **SVM*** files.
 - c. Click **Add Source**.
 - d. From the *Field Selector* tab of the **Data Explorer**, notice that all three bands (SVM03, SVM04, and SVM05) are included in the list in the **Fields** panel.
7. Use the *VIIRS_M_RGB(M5, M4, M3)* formula to create an RGB display from the data source.
 - a. Add a new one-paneled map display by selecting *File -> New Display Tab -> Map Display -> One Panel* menu item. Close any previously-existing tabs.
 - b. In the *Field Selector* tab of the **Data Explorer**, select **Formulas**.
 - c. Choose the *VIIRS_M_RGB(M5, M4, M3)* formula, the *RGB Composite* display type, and click **Create Display**.
 - d. In the **Field Selector** window, select the following:

i. **Field M5: VIIRS 2019-09-02 18:15:29 GMT -> Image -> VIIRS-M5-SDR_All/Radiance.**

Once the **Region** tab appears, use *Shift+Left-Click+Drag* to draw a bounding box within the bounds of the data. This subsetting step only has to be done for this field (the Red component of the formula) since this domain will be carried over to the Green and Blue fields. Make sure that the subsetting area includes land, water, and data near the east and/or west edges of the granule to include the bowtie deletion regions.

ii. **Field M4: VIIRS 2019-09-02 18:15:29 GMT -> Image -> VIIRS-M4_SDR_All/Radiance.**

iii. **Field M3: VIIRS 2019-09-02 18:15:29 GMT -> Image -> VIIRS-M3_SDR_All/Radiance.**

e. Once all three **Region** subset tabs are visible, click **OK** at the bottom of the **Field Selector** window.

8. Investigate and brighten the display to make it easier to interpret.

a. From the **Main Display** window, notice that the display looks mainly black and white, though the bowtie deletion lines have been removed and the data is at full (750 meter) resolution.

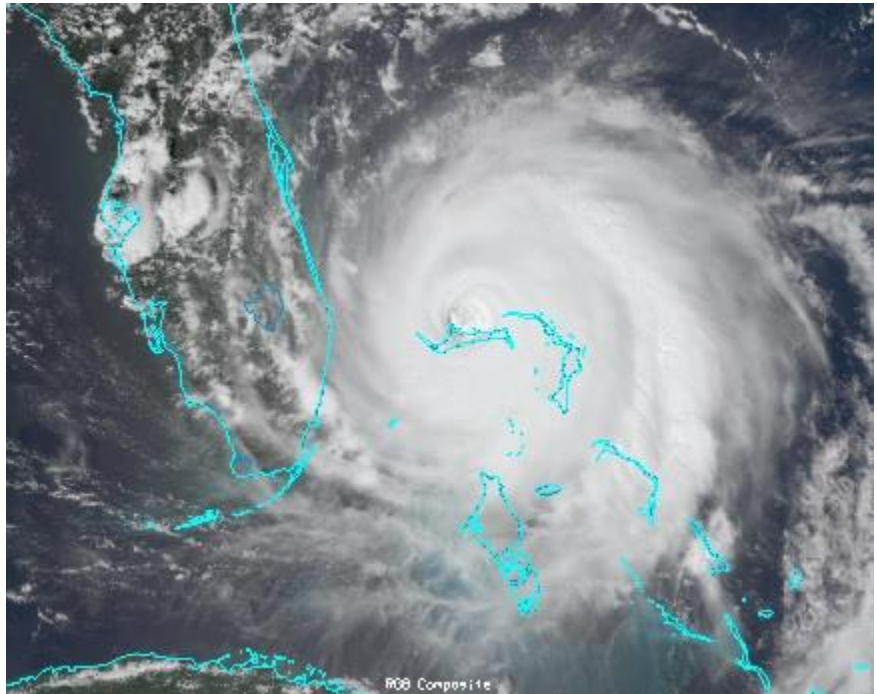
b. Click on the blue **RGB Composite** text in the **Legend** of the **Main Display** to get to the **Layer Controls** tab of the **Data Explorer**.

c. The intensities of the red, green, and blue colors are controlled by their Gamma values. Adjust all three colors to have a Gamma value of 0.4 by typing *0.4* into the **Common Gamma** field of the **Layer Controls** and click **Apply to All Gamma Fields**.

d. Go back to the **Main Display** and notice how the red, green, and blue colors are now more intense and you can see green and brown over the land and blue over areas of water.

e. By default, a wireframe box is drawn in the **Main Display**. This is the white box going around and partially through the data. Disable the wireframe box by unselecting **View -> Show -> Wireframe Box** from the panel in the **Main Display** window.

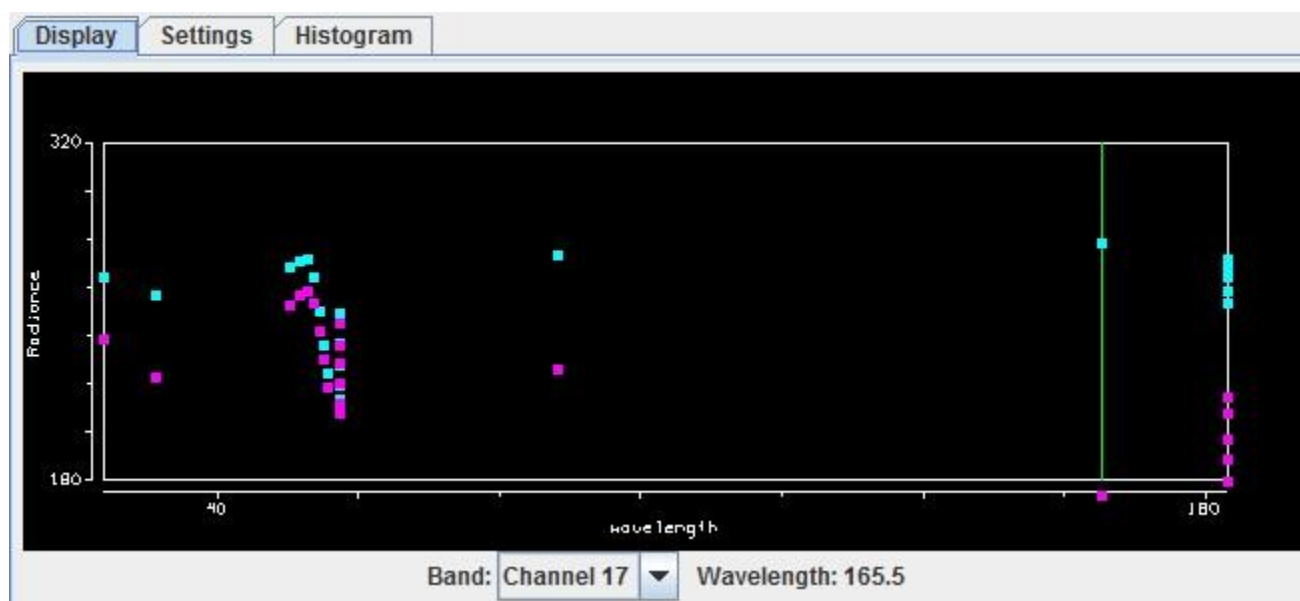
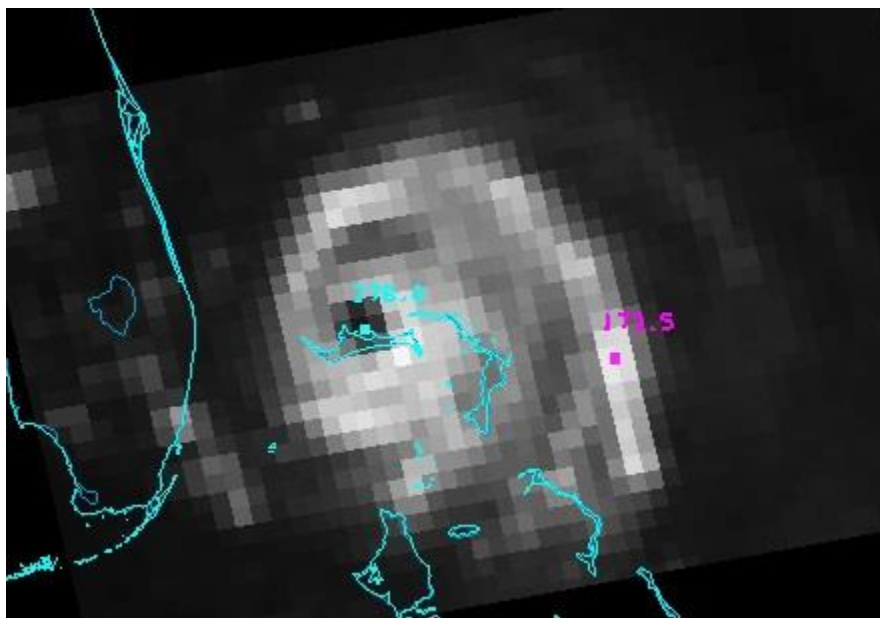
f. When you are done investigating the display, remove all layers and data sources.



Note that this formula suggests the user selects SVM05 for Red, SVM04 for Green, and SVM03 for Blue to create a true color RGB display. However, any bands can be used to create different types of RGB displays.

Exercise

From the JPSS chooser, select all four GATMO-SATMS* files and click Add Source. Create a MultiSpectral Display of the Brightness Temperature field subsetting over the hurricane. Investigate how changing the channel in the Layer Controls impacts the display. Attempt to find a channel that makes the areas of heaviest precipitation stand out well. For example:



Under Development – Orbit Tracks


Introduction

McIDAS-V has the ability of displaying TLE (Two Line Elements) Satellite Orbit Track data through the *Under Development -> Satellite Orbit Track* chooser. TLE data can be loaded into the Field Selector in three ways:

- Through a local file on your machine or a mounted drive
- Through ADDE
- Through a URL

This tutorial will describe how to load data through a local file and a URL. Different options for customizing the display will be covered as well.

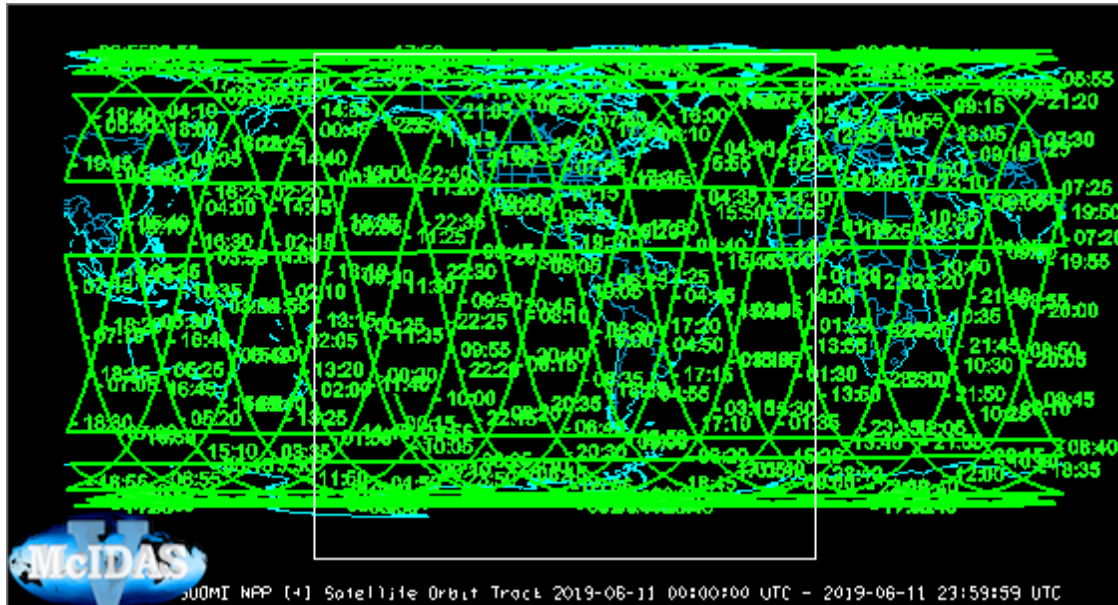
Displaying TLE Data from a Local File

1. In the **Main Display**, remove all layers and data sources via the *Edit -> Remove -> All Layers and Data Sources* menu item.
2. Create a new tab with the *File -> New Display Tab -> Map Display -> One Tab* menu item.
3. Create a data source of satellite track data from a local file.
 - a. In the **Data Sources** tab of the **Data Explorer**, navigate to the *Under Development – Satellite Orbit Track* chooser.
 - b. Under **Local**, verify that *File* is selected.
 - c. Navigate to the *<local path>/Data/SatelliteTracks* directory and select *noaa.txt*.
 - d. Click **Add Source**.
4. Display a full day's worth of Suomi NPP tracks from September 18, 2019.
 - a. In the **Fields** panel of the **Field Selector**, select the *SUOMI NPP [+]* field.
 - b. In the **Displays** panel, select *Satellite Orbit Track*.
 - c. The **Time Range** panel allows for selecting the beginning and ending date/time of the track to plot. McIDAS-V will default to displaying the tracks from 00:00:00Z – 23:59:59Z on the current date. To display the full track from September 18, 2019, click on the calendar widget () next to **Begin Date** and select *Sep 18, 2019*. Repeat this process for the **End Date** field.

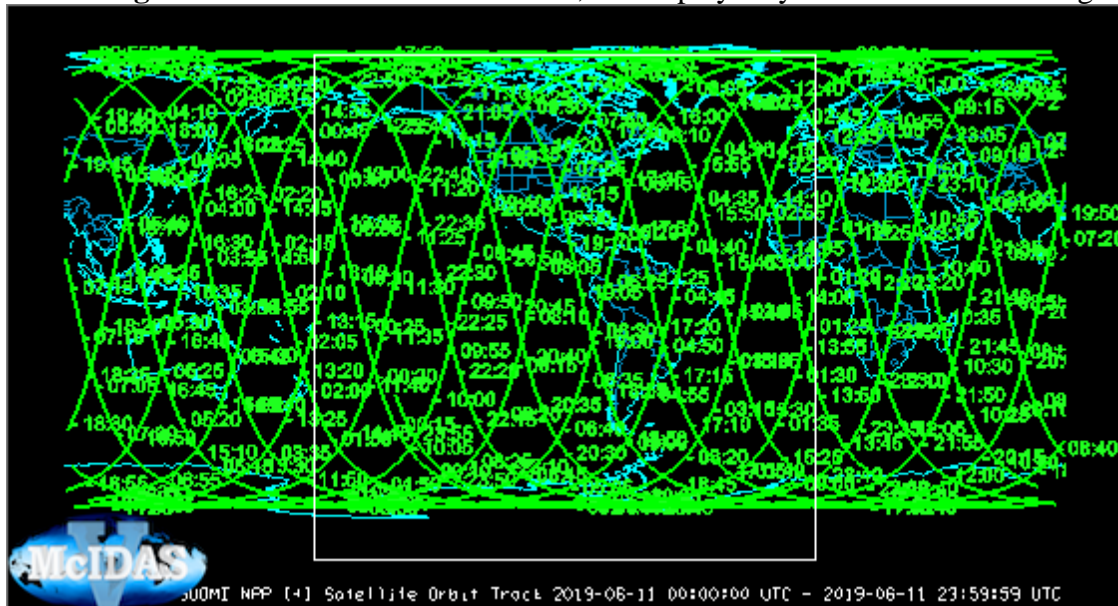
d. Click **Create Display**.

5. Inspect the display of the Suomi NPP tracks.

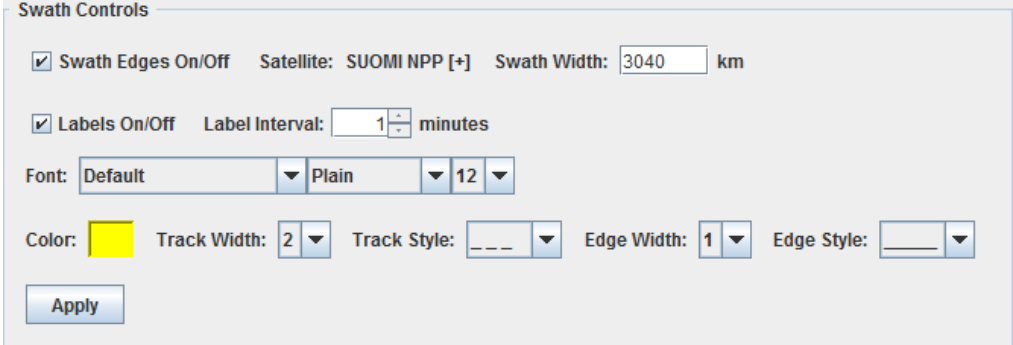
a. When the display is created, you may see horizontal lines going across the display, as is shown here:



If you see these horizontal lines, go to the **Layer Controls** tab of the **Data Explorer** for the orbit track layer and select **Edit -> Properties**. At the bottom of the **Properties** window, uncheck **Use Fast Rendering** and click **OK**. Once this is done, the display may look like the following image:

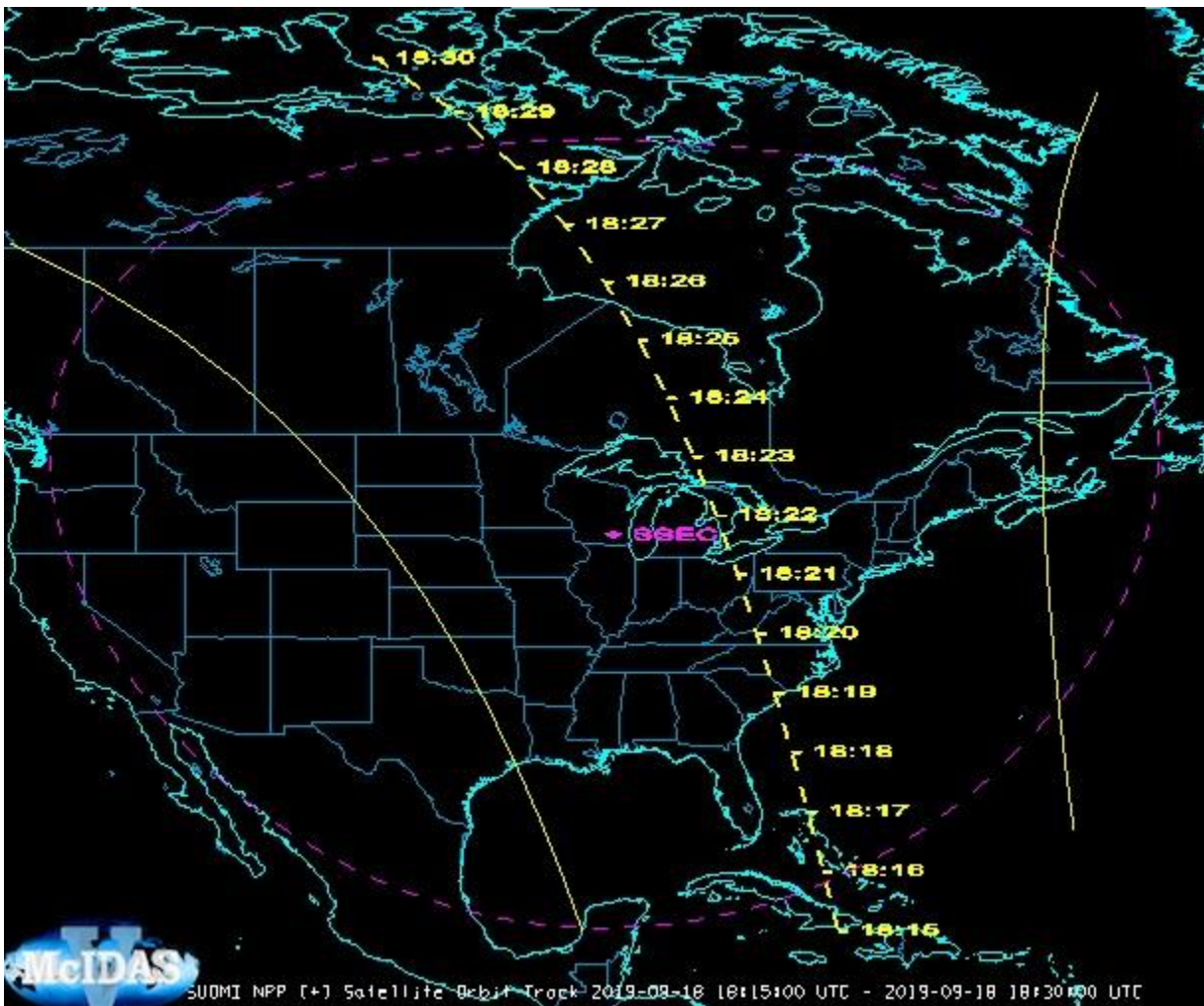


There are times where disabling fast rendering can remove spurious lines. There are other times where enabling fast rendering can fix gaps in lines when viewing a polar projection. In general, if the display's lines don't look correct, changing the fast rendering setting for the layer is a good idea.

- b. Zoom in on the region of interest in the **Main Display** by using *Shift+Left-Click+Drag*. For example, select an area over the central United States. Observe the track times, and determine a good time range where the track lines cross through the United States. The times are written along the track. For example, you may determine a time range of 18:15 through 18:30 is good. In the next section of this tutorial, this time range will be plotted.
6. Create a new tab via the *File -> New Display Tab -> Map Display -> One Panel* menu item.
 7. Display Suomi NPP satellite tracks for the time range chosen in step 5b.
 - a. From the *Field Selector* tab of the **Data Explorer** select *SUOMI NPP [+]*.
 - b. In the **Time Range** panel, verify that *Sep 18, 2019* is selected for the **Begin Date** and **End Date** fields. For **Begin Time**, enter *18:15:00* (or the start time determined in step 5b) and for **End Time** enter *18:30:00* (or the end time determined in step 5b) and click **Create Display**.
 8. Modify different aspects of the track line display.
 - a. Zoom in over the track data in the **Main Display** by using *Shift+Left-Click+Drag*.
 - b. Add swath edge lines to the display by going to the *Layer Controls* and enabling **Swath Edges On/Off**. Enter a value of 3040 into the **Swath Width** field and click **Apply**.
 - c. By default, time labels are drawn along the track every 5 minutes. Change the display to show time labels at every minute by going to the *Layer Controls*, setting **Label Interval** to *1*, and clicking **Apply**.
 - d. From the *Layer Controls*, change the color of the lines and labels in the display to yellow by clicking on the green **Color Swatch**, selecting *Yellow*, clicking **OK** in the **Color** window, and clicking **Apply**.
 - e. From the *Layer Controls*, change the **Track Style** from a solid line to a dashed line by using the dropdown menu and click **Apply**.
 
 - f. From the *Layer Controls*, change the **Edge Style** from a dashed line to a solid line by using the dropdown menu and click **Apply**.

9. Add a SSEC (Space Science and Engineering Center) ground station to the display.
 - a. From the **Ground Stations** panel of the *Layer Controls*, select *SSEC* for the **Ground Stations Available** menu and click **Add Selected**.

Investigate the display. A ground station for SSEC has been added to the display over south central Wisconsin. There is a ring around the ground station that denotes the region where an antenna at the SSEC ground station can detect the Suomi NPP satellite. The size of this ring is based on the altitude of the satellite as well as the **Antenna Angle**, which is defined at the bottom of the *Layer Controls*. This is the angle that the antenna at the SSEC ground station scans at. Once the ground station is added, the final display may look like:



Bundles

Unzipped Bundles

1. Remove all layers and data sources, create a new one-paneled map display, and close any previously-existing tabs.
2. From the *Satellite* -> *Imagery* chooser of the **Data Explorer**, create a data source of the most recent GOES-16 CONUS Cloud Top Pressure level 2 product.
 - a. Connect to *lead.unidata.ucar.edu/NPGOESR*.
 - b. In the **Image Type** dropdown, select *CTPRCN – GOES-East Level2 CONUS Cloud Top Pressure Product*.
 - c. From the *Relative* tab of the **Times** panel, enter *1* into the **Number of times** field.
 - d. Click **Add Source**.
3. Display the Cloud Optical Depth data.
 - a. From the **Fields** panel of the *Field Selector*, select *L2: CTP – Cloud Top Pressure – Ctp*.
 - b. From the **Displays** panel, select *Imagery -> Image Display*.
 - c. From the *Advanced* subset tab, click the four arrow green button to get the full resolution of the data.
 - d. Click **Create Display**.

Note that the display automatically uses a parameter default to set the appropriate enhancement and range for the product.
4. Save a bundle of the display.
 - a. From the **Main Display** window, select *File -> Save Bundle*.
 - b. In the **Save** window, navigate to the directory where you want to save the bundle to.
 - c. Enter a name of the file in the **File Name** field.
 - d. In the **Files of Type** dropdown menu, choose *McIDAS-V Bundles (*.mcv)*.
 - e. Click **Save**.
5. Load the bundle.

- a. Remove all layers and data sources, create a new one-paneled map display, and close any previously-existing tabs.
- b. From the **Main Display** window, select *File -> Open File*.
- c. In the **Open File** window, navigate to the location where the bundle was saved to, select the bundle, and click **Open**.

With this being an unzipped bundle, no actual data is saved in the bundle file, just a reference to the data. When the bundle is loaded, McIDAS-V will go back to the same server used to create the bundle and pull in the most recent version of the data. The data will be displayed in the same way that it was when the bundle was saved.

Bundles like this can be useful to create for forecasting. For example, a forecaster can create a session containing several tabs of radar, grid, point, and satellite data that he/she wants to use again in the future. Once this bundle is saved, it's easy to restore the same setup that was used previously. Bundle files can also be passed around to other McIDAS-V users.

Default Bundles

1. Remove all layers and data sources, create a new one-paneled map display, and close any previously-existing tabs.
2. From the *Satellite -> Imagery* chooser of the **Data Explorer**, create a data source of the most recent GOES-16 CONUS band 14 data.
 - a. Connect to *lead.unidata.ucar.edu/RTGOESR*.
 - b. In the **Image Type** dropdown, select *CONUSC14 – GOES-East CONUS 11.2 um IR Imagery/SST/clouds/rainfall*
 - c. From the *Relative* tab of the **Times** panel, enter *1* into the **Number of times** field.
 - d. Click **Add Source**.
3. Display the band 14 temperature data.
 - a. From the **Fields** panel of the *Field Selector*, select *11.2 um IR Imagery,SST,clouds,rainfall -> Temperature*.
 - b. From the **Displays** panel, select *Imagery -> Image Display*.
 - c. From the *Advanced* subset tab, click the four arrow green button to get the full resolution of the data.
 - d. Click **Create Display**.
4. Save the display as a favorite bundle.

- a. From the **Main Display** window, select *File -> Save Favorite*.
 - b. In the **Save As Favorite** window, make sure that *General* is chosen for **Category**.
 - c. In the **Name** field, enter a name, such as: *GOES16_band14*.
 - d. Click **OK**.
5. Set the favorite bundle that was just saved to be the default bundle, meaning it will be loaded right away when McIDAS-V is started up.
- a. From the **Main Display** window, select *Edit -> Preferences*.
 - b. In the **User Preferences** window, open the *Advanced* tab.
 - c. Next to **Defaults**, enable the **Specify default bundle** option.
 - d. Click the **Browse** button next to the **Defaults** text field.
 - e. From the **Open** window, go to the *General* directory, select the bundle, and click **Open**.
 - f. Click **OK** to close the **User Preferences** window.
6. Verify that the default bundle works as expected.
- a. Exit McIDAS-V.
 - b. Restart McIDAS-V.
 - c. Optionally, disable the default bundle so it doesn't load every time McIDAS-V starts. To do this, go back to the *Advanced* tab of the **User Preferences** window, disable **Specify default bundle**, and click **OK**.

Toolbar Bundles

1. Remove all layers and data sources, create a new one-paneled map display, and close any previously-existing tabs.
2. From the *Point Observations -> Plot/Contour* chooser of the **Data Explorer**, create a data source of the most recent surface hourly observation data.
 - a. Connect to *adde.ssec.wisc.edu/RTPTSRC*.
 - b. In the **Point Type** dropdown, select *SFCHOURLY – Real-Time SFC Hourly*

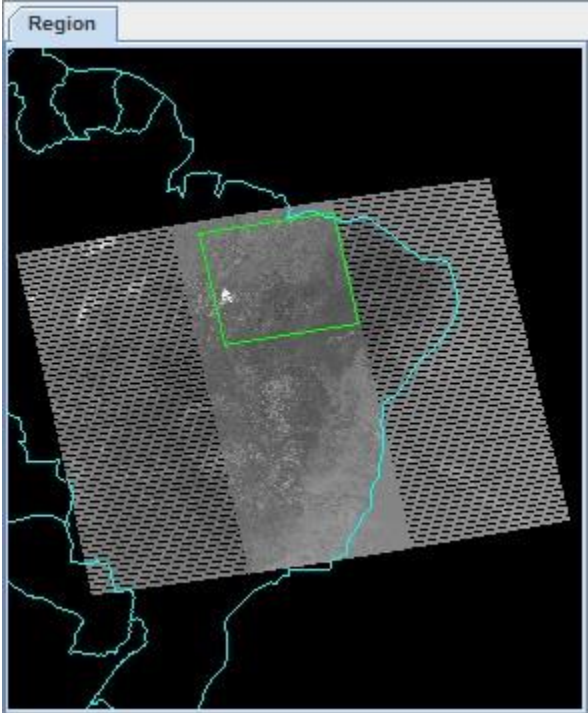
- c. From the *Relative* tab of the **Times** panel, enter *1* into the **Number of times** field.
 - d. Click **Add Source**.
3. Display the surface hourly data.
 - a. From the **Fields** panel of the *Field Selector*, select *Point Data*.
 - b. From the **Displays** panel, select *Point Data -> Point Data Plot*.
 - c. Click **Create Display**.
 4. Save the display as a toolbar bundle.
 - a. From the **Main Display** window, select *File -> Save Favorite*.
 - b. In the **Save As Favorite** window, enter *Toolbar>Point* for **Category**.
 - c. In the **Name** field, enter a name, such as: *World METARS*.
 - d. Click **OK**.
 5. Verify that the toolbar bundle works as expected.
 - a. Remove all layers and data sources, create a new one-paneled map display, and close any previously-existing tabs.
 - b. From the **Main Toolbar** in the **Main Display** window, select *Point -> World METARS*.
 - c. If prompted, choose to replace the session.

Toolbar bundles are an easy way to customize your session to make frequently displays readily accessible in an organized way. In this example, the bundle was saved under a category called “Point”. Bundles can be added directly to the toolbar as well by simply specifying “Toolbar” as the Category when saving the bundle. An example of how multiple bundles can be saved to the toolbar can be seen under the Current WX button in the toolbar, which contains a variety of unzipped bundles for different types of data (grid, point, and satellite).

- d. Toolbar buttons can also be accessed through the **Local Favorite Bundles Manager** window, which can be accessed through the *Bundles -> Manage* menu item in the **Main Display**. This window gives the ability to load favorite bundles, remove them, or organize them.

Zipped Bundles

1. Remove all layers and data sources, create a new one-paneled map display, and close any previously-existing tabs.

2. From the *Under Development* -> *Imagery* - *JPSS* chooser of the **Data Explorer**, create a data source of Suomi NPP band 13 data.
 - a. Navigate to the *<local path>/Data/JPSS/Fire* directory.
 - b. Select the *SVM13** file.
 - c. Click **Add Source**.
3. Display the brightness temperature data at full resolution over a portion of the data containing fire pixels.
 - a. In the **Fields** panel of the *Field Selector*, select *Image* -> *VIIRS-M13-SDR_All/BrightnessTemperature* field.
 - b. In the **Displays** panel, select *Imagery* -> *Image Display*.
 - c. In the **Region** tab, use *Shift+Left-Click+Drag* to subset a region in the northern part of the data between the bowtie deletion lines.
 
 - d. Click **Create Display**.
4. Modify the enhancement range to make the fire pixels easier to locate.
 - a. Zoom in on the region where the data is displayed in the **Main Display**.
 - b. *Right-Click* on the grayscale enhancement in the **Legend** and choose *Change Range*.
 - c. In the **Change Range** window, change the **To** value to *340* and click **OK**.
5. The fire pixels are a little easier to see after the previous step. Modify the enhancement itself to make them stand out even more.
 - a. *Right-Click* on the enhancement in the **Legend** and choose *Edit Color Table*.
 - b. The **Color Table Editor** window works by interpolating colors between breakpoints. Add a new breakpoint at *330* by *Right-Clicking* above the enhancement and choosing *Add Breakpoint* -> *At Data Point*. In the **Breakpoint Value** window, enter a value of *330* and click **OK**.
 - c. With the *330* breakpoint active (yellow outline around triangle at breakpoint), use the **HSV** tab to make the breakpoint yellow.

- d. *Left-Click* on the 340 breakpoint to make it active, and set this breakpoint to red using the same process to set the 330 breakpoint to yellow.
 - e. *Right-Click* on the 340 breakpoint's triangle and select **Edit Colors -> Interpolate -> Left**. This will make the upper end of the enhancement shade from yellow to red.
 - f. Investigate the display to see how the fires are now easier to see.
6. Save the display as a zipped bundle.
 - a. From the **Main Display** window, select **File -> Save Bundle**.
 - b. In the **Save** window, navigate to the directory where you want to save the bundle to.
 - c. Enter a name of the file in the **File Name** field.
 - d. In the **Files of Type** dropdown menu, choose *McIDAS-V Zipped Data Bundles (*.mcvz)*.
 - e. Click **Save**.
 - f. In the **Save Data** window, select *Save All Displayed Data* and click **OK**. Wait for the **Creating Zipped Bundle** window to go away, meaning that the bundle has finished writing itself.
 7. Load the bundle.
 - a. Remove all layers and data sources, create a new one-paneled map display, and close any previously-existing tabs.
 - b. From the **Main Display** window, select **File -> Open File**.
 - c. In the **Open File** window, navigate to the location where the bundle was saved to, select the bundle, and click **Open**.

Zipped bundles contain the data within the file itself. These types of bundles can be useful for case studies, where you don't want to load in the most recent version of the data, but the data for a particular date/time of interest. Just like unzipped bundles, these zipped bundles can be sent to other users of McIDAS-V as well.

Saving Images and Animations

Single-Paneled Image

1. Remove all layers and data sources, create a new one-paneled map display, and close any previously-existing tabs.
2. From the *Satellite* -> *Imagery* chooser of the **Data Explorer**, create a data source of the most recent GOES-16 CONUS band 14 data.
 - a. Connect to *lead.unidata.ucar.edu/RTGOESR*.
 - b. In the **Image Type** dropdown, select *CONUSC14 – GOES-East CONUS 11.2 um IR Imagery/SST/clouds/rainfall*
 - c. From the *Relative* tab of the **Times** panel, enter *1* into the **Number of times** field.
 - d. Click **Add Source**.
3. Display the band 14 temperature data.
 - a. From the **Fields** panel of the *Field Selector*, select *11.2 um IR Imagery,SST,clouds,rainfall -> Temperature*.
 - b. From the **Displays** panel, select *Imagery -> Image Display*.
 - c. Click **Create Display**.
4. Capture an image of the display.
 - a. From the display panel in the **Main Display**, select *View -> Capture -> Image*.
 - b. In the **Save** window, select the directory for where to save the image.
 - c. Under **Capture What**, select *Current View*.
 - d. Under **Quality**, select *High*.
 - e. In the **File Name** field, enter a name followed by “.jpg”. For example, *Band_14.jpg*.
 - f. Click **Save**.
5. Outside of McIDAS-V, open the captured image in a software that allows you to view JPG images. For example, an internet browser.

Single-Paneled Animation

1. Remove the currently-displayed layer in the display window.
2. Display the 5 most recent band 14 CONUS images.
 - a. From the *Data Sources* tab of the **Data Explorer** window, enter a value of 5 into the **Number of times** field and click **Add Source**.
 - b. From the **Fields** panel of the *Field Selector*, select *11.2 um IR Imagery,SST,clouds,rainfall -> Temperature*.
 - c. From the **Displays** panel, select *Imagery -> Image Display*.
 - d. Click **Create Display**.
3. Capture a movie of the display.
 - a. From the display panel in the **Main Display**, select *View -> Capture -> Movie*.
 - b. In the **Movie Capture** window, verify that *Current View* is selected under **What to capture**.
 - c. Click the **Time Animation** button. Note that the **Main Display** will now step through the loop, capturing an image of each frame. While this process is occurring, don't interact with the display or it's possible that the movie might not save correctly.
 - d. In the **Save** window, enter *1* for both **Frames per second** and **End Frame Pause**. Navigate to the directory where you want to save the animation. Enter a file name, choose *Animated GIF (*.gif)* for **Files of Type**, and click **Save**.
 - e. Close the **Movie Capture** window.
4. Outside of McIDAS-V, open the saved animation in a software that allows you to view animated GIFs. For example, an internet browser.

Multi-Paneled Image

1. Remove all layers and data sources.
2. In the **Main Display** window, select *File -> New Display Tab -> Map Display -> Four Panels*.
3. Remove any old tabs from previous exercises.
4. From the *Satellite -> Imagery* chooser of the **Data Explorer**, create a data source of the most recent GOES-16 CONUS data.

- a. Connect to *lead.unidata.ucar.edu/RTGOESR*.
 - b. In the **Image Type** dropdown, select *CONUS – GOES-East CONUS all bands*.
 - c. From the **Relative** tab of the **Times** panel, enter *1* into the **Number of times** field.
 - d. Click **Add Source**.
5. Display band 1 data in panel 1 of the **Main Display** window.
- a. In the **Main Display** window, *Left-Click* in the upper left panel (Panel 1) to make it active. The active panel is the one with the blue border around it. Any time **Create Display** is clicked in the **Field Selector**, the display will go to the active panel.
 - b. In the **Fields** panel of the **Field Selector**, select *0.47 um VIS aerosol-over-land -> Brightness*.
 - c. In the **Displays** panel, select *Imagery -> Image Display*.
 - d. Click **Create Display**.
6. Repeat this process for the other three panels, displaying any band/calibration of your choosing.
7. Capture an image of the display.
- a. From the active display panel in the **Main Display**, select *View -> Capture -> Image*.
 - b. In the **Save** window, select the directory for where to save the image.
 - c. Under **Capture What**, select *All Views*.
 - d. Under **Quality**, select *High*.
 - e. In the **File Name** field, enter a name followed by “.jpg”. For example, *ABI_4_panel.jpg*.
 - f. Click **Save**.
8. Outside of McIDAS-V, open the captured image in a software that allows you to view JPG images. For example, an internet browser.

Data Polling

McIDAS-V has the ability to automatically update displays at a user-defined time interval. Every “x” number of minutes, McIDAS-V will check the data server to see if new data is available. If new data exists, the display will be updated. Below is an example of data polling with GOES-17 mesoscale-1 sector data, which has a 1-minute frequency.

1. Remove all layers and data sources, create a new one-paneled map display, and close any previously-existing tabs.
2. Add a new remote dataset for GOES-17 data.

- a. From the **Main Display** window, select **Tools -> Manage ADDE Datasets**.
- b. From the **Remote Data** tab of the **ADDE Data Manager**, click **Add New Dataset**.
- c. In the **Add Remote Dataset** window, enter the following:

Server: *adde.ucar.edu*

Dataset: *RTGOESS*

Dataset Types: *Image*

- d. Click **Verify and Add Server**.
- e. Click **Ok** to close the **ADDE Data Manager** window.

3. From the **Satellite -> Imagery** chooser of the **Data Explorer**, create a data source of the 5 most recent GOES-17 mesoscale-1 sector band 14 data.


- a. Connect to *adde.ucar.edu/RTGOESS*.
- b. In the **Image Type** dropdown, select *MIC14 – GOES-West Mesoscale-1 11.2 um IR Imagery/SST/clouds/rai*.
- c. From the **Relative** tab of the **Times** panel, enter 5 into the **Number of times** field.
- d. Click **Add Source**.

4. Set up the data polling.
 - a. From the *Field Selector* tab of the **Data Explorer**, *Right-Click* on the **M1C14 – GOES-West** data source and select *Properties*.
 - b. Enable the **Active** box next to *Automatically Reload*. This is the switch that turns polling on for the data source.
 - c. Enter a value of *1* into the **Check Every** box to have McIDAS-V check for new data every minute.
 - d. Enable **Reload Displays** at the bottom of the window. This toggles the display to update.
 - e. Click **OK** and close the **Properties** window.
5. Display the band 14 Temperature data.
 - a. From the **Fields** panel of the *Field Selector*, select *11.2 um IR Imagery,SST,clouds,rainfall -> Temperature*.
 - b. From the **Displays** panel, select *Imagery -> Image Display*.
 - c. Click **Create Display**.
6. Investigate how the data polling works. Right when the data displays, take a note of the data times being displayed. After about a minute, check and see if new data is loaded in. Note that the display still has 5 timesteps. This means that newer times were added to the end of the loop, and the older times were removed from the beginning of the loop.

Different Types of Display Panels

Globe Panel

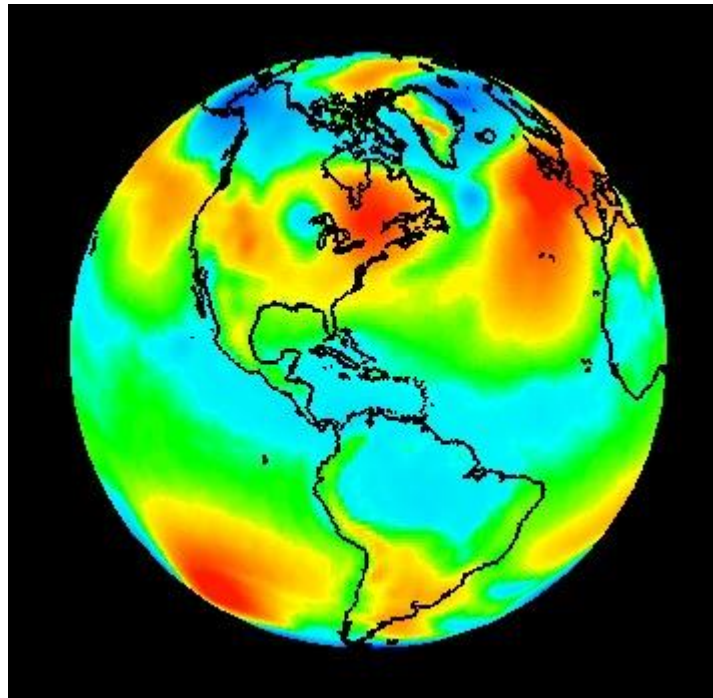
1. Set up the display window there is only one tab containing a globe panel.
 - a. Remove all layers and data sources.
 - b. From the **Main Display** window, select **File -> New Display Tab -> Globe Display -> One Panel**.
 - c. Close any other previously-existing displays.
2. If you are using real-time data, skip to step 3. Otherwise, load the file `<local path>/Data/Grid/GFS_Global_1p0deg_MSLP.nc`.
 - a. In the **Data Explorer**, navigate to the **Gridded Data -> Local** chooser.
 - b. Select the `<local path>/Data/Grid/GFS_Global_1p0deg_MSLP.nc` file.
 - c. Click **Add Source**.
3. If you are using realtime data, follow the instructions below.
 - a. Select the **Gridded Data -> Remote** chooser from the **Data Sources** tab of the **Data Explorer**.
 - b. From the **Catalog** pull down menu, select `http://www.ssec.wisc.edu/mcidas/software/v/threddsRTModels.xml`.
 - c. Add the **Realtime Data -> NCEP Model Data -> Global Forecast System (GFS) -> GFS One Degree – Global Coverage -> GFS One Degree Analysis -> Latest** source.
4. Display the MSL Pressure.
 - a. In the **Field Selector**, select the **2D grid -> Mass -> Pressure reduced to MSL @ Mean sea level** field if using the remote data source. If using the local file, select the **Pressure reduced to MSL @ Mean sea level** field.
 - b. Select the **Plan Views -> Color-Shaded Plan View** Display.
 - c. In the **Times** tab, use the dropdown to choose **Use Selected**, and select the first six times.
 - d. Click **Create Display**.
5. Adjust and inspect the display.
 - a. From the **Layer Controls** tab of the **Data Explorer**, enable **Shade Colors**.

- b. In the **Main Display**, disable *View -> Show -> Wireframe Box*.
- c. Use *Right-Click+Drag* to rotate the globe. Also, the auto-rotate button () in the navigation toolbar can automatically rotate the globe.
- d. Click on the blue **Default Background Maps** text in the **Legend** to get to the *Layer Controls* for the maps. Do the following to make the map lines easier to see:

Disable the checkbox for **North & Central America**.

Disable the checkbox for **World Political Boundaries**.

For **World Coastlines**, change the color to black and use the dropdown menu to change the line width from *1.0* to *2.0*.



Transect and Map Panel


1. Set up the display window there is only one tab containing a transect and a map.
 - a. Remove all layers and data sources.
 - b. From the **Main Display** window, select *File -> New Display Tab -> Misc -> Transect And Map*.
 - c. Close any other previously-existing displays.

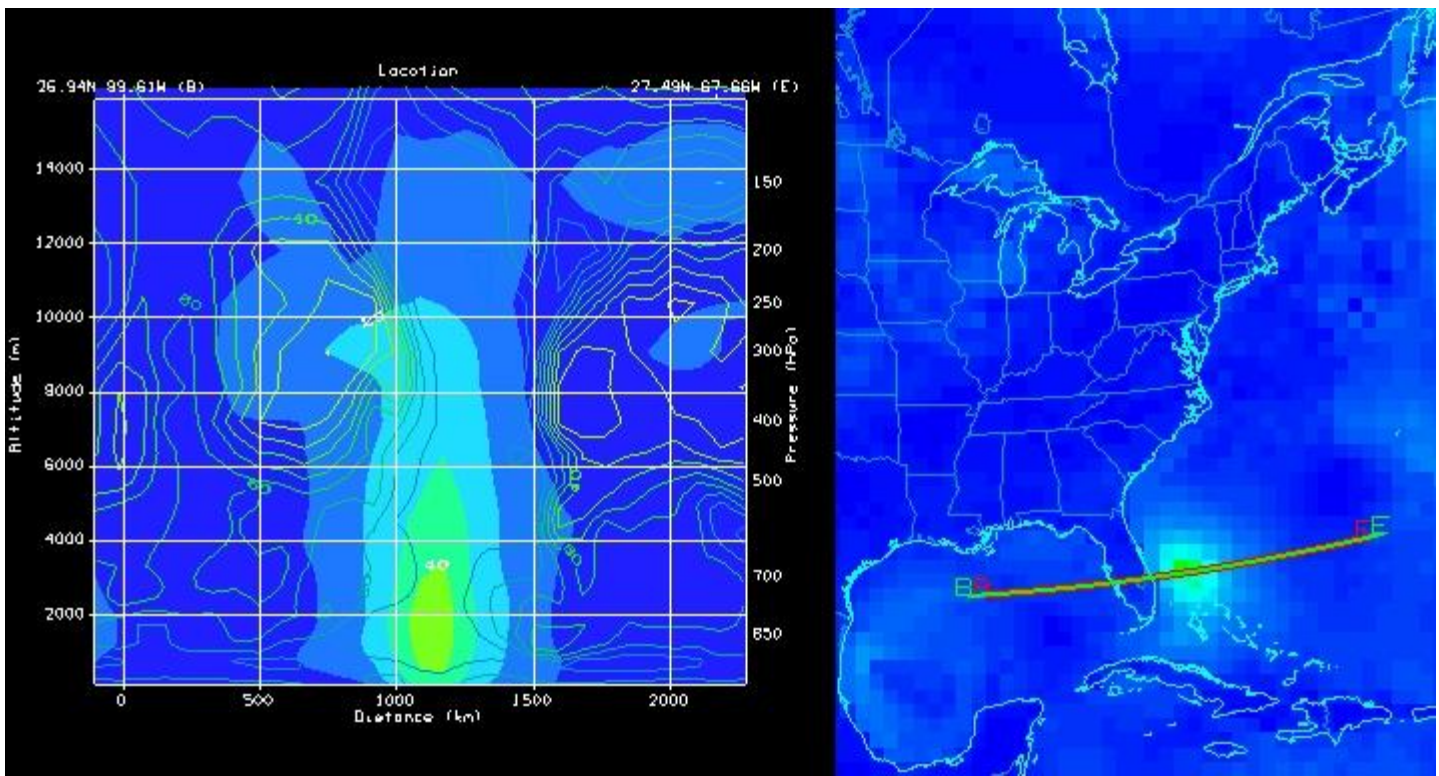
2. Load the file *<local path>/Data/Grid/GFS_CONUS_80km_20190903_1200.grib1*.
 - a. In the **Data Explorer**, navigate to the *Gridded Data -> Local* chooser.
 - b. Select the *<local path>/Data/Grid/GFS_CONUS_80km_20190903_1200.grib1* file.
 - c. Click **Add Source**.

3. Add a display of wind speed data at the surface for the first timestep to the right map panel in the **Main Display**.
 - a. In the **Main Display**, *Left-Click* in the right map panel to make it active.
 - b. In the **Fields** panel *Field Selector*, select the *2D -> Derived -> Speed* field.
 - c. In the **Displays** panel, select *Plan Views -> Color-Shaded Plan View*.
 - d. In the **Times** tab, change the dropdown to *Use Selected* and choose the first timestep only.
 - e. Click **Create Display**.

4. Add a vertical cross section of wind speed to the left transect panel in the **Main Display** window.
 - a. In the **Main Display**, *Left-Click* in the left transect panel to make it active.
 - b. In the **Fields** panel of the *Field Selector*, select the *3D -> Derived -> Speed* field.
 - c. In the **Displays** panel of the *Field Selector*, select *Vertical Cross Sections -> Color-Filled Contour Cross Section*.
 - d. In the **Times** tab, change the dropdown to *Use Selected* and choose the first timestep only.
 - e. Click **Create Display**.

5. A vertical cross section of wind speed should now be shown in the left panel in the **Main Display**. This cross section is drawn using the location of the transect line in the right map panel. Change the location of this transect line so it crosses over Hurricane Dorian.

- a. In the left transect panel of the **Main Display**, select the *Transects -> Edit* menu item. This opens a **Transect Drawing Control** window.
 - b. From the **Controls** tab of the **Transect Drawing Control** window, select the *Create A Transect* () **Mode** option.
 - c. In the right map panel in the **Main Display**, use *Left-Click+Drag* to draw a transect line that crosses over Hurricane Dorian. At this point, you should notice that the left transect panel updates to showing a vertical cross section of wind speeds over the hurricane.
6. The transect panel can overlay multiple parameters together at once. For example, add a vertical cross section of relative humidity to the transect panel.
- a. *Left-Click* in the left transect panel of the **Main Display** to make it active.
 - b. From the **Fields** panel of the *Field Selector*, select the *3D grid -> Relative humidity @ Isobaric surface* field.
 - c. From the **Displays** panel, select *Vertical Cross Sections -> Contour Cross Section*.
 - d. Verify that only the first time is selected in the *Times* tab.
 - e. Click **Create Display**.



Creating a Derived Field

1. Remove all layers and data sources from the previous displays. Create a new, one-paneled map display and close any previously-existing displays.
2. Add a gridded data source. If you are using real-time data, skip to step 3.
 - a. In the *Data Sources* tab of the **Data Explorer**, navigate to the *Gridded Data -> Local* chooser.
 - b. Select the *<local path>/Data/Formulas/GFS_CONUS_80km_20190903_1200.grib1*.
 - c. Click **Add Source**.
3. Add a remote GFS 80 km CONUS data source.
 - a. In the *Data Sources* tab of the **Data Explorer**, navigate to the *Gridded Data -> Remote* chooser.
 - b. Select *Realtime Data -> NCEP Model Data -> Global Forecast System (GFS) -> GFS-CONUS 80km -> Latest*.
 - c. Click **Add Source**.
4. Inspect the 2D native and derived fields.
 - a. Expand the *2D grid* tree, and all of the fields listed directly under 2D grid are native fields, or fields included with the data.
 - b. Expand the *2D grid -> Derived* tree to see all of the derived fields. These derived fields are created by formulas. The majority of the formulas from these derived fields can be found by *right-clicking* on **Formulas** in the *Field Selector* and navigating to *Edit Formulas -> Derived Quantities*.
5. Create a new derived quantity to inspect the difference between Pressure and MSLP.
 - a. In the *Field Selector*, *right-click* on **Formulas** and select *Create Formula*.
 - b. At the top of the **Formula Editor** window, enter the following:
 - i. Description: *MSLP – pressure (from %N1% & %N2%)*
 (1) This is how the derived field will list in the *Field Selector*. The ‘N1’ and ‘N2’ will allow for the name of the derived field to include the names of the two fields that it was derived from: **MSLP**

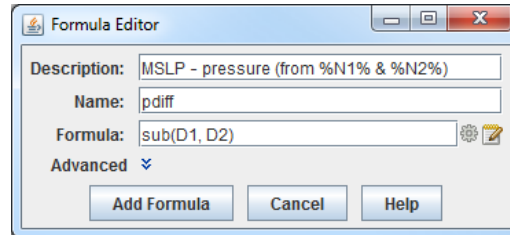
– *pressure (from Pressure_reduced_to_MSL_msl & Pressure_surface)*

ii. Name: *pdiff*

(1) This is how the derived field can be referred to in a parameter default

iii. Formula: *sub(D1, D2)*

(1) This is the actual formula. D1 and D2 are used as placeholders for the two fields to be operated on. These fields will be defined in step 3d.



c. Expand **Advanced**.

d. In the *Derived* tab, do the following:

i. Uncheck **For end user** and check **Create derived quantities**.

(1) Unchecking **For end user** keeps the formula from being listed out with the rest of the formulas in the *Field Selector*. Checking **Create derived quantities** creates a derived quantity (field) from this formula.

(2) In the **Parameters** menu, *Right-Click* in the text box, navigate through the menu tree to select the *Pressure_reduced_to_MSL_msl* field, type a comma (,), and *Right-Click* again to choose the *Pressure_surface* field. These fields are the **D1** and **D2** specified in the formula (step 5b iii above).

(3) The **Categories** menu here doesn't have to be used since specific field names are being selected. If we were using an alias, such as **PRESSURE**, then this **Categories** menu could be used to tell McIDAS-V to only look for 2D gridded pressure fields by entering *GRID-2D-**.

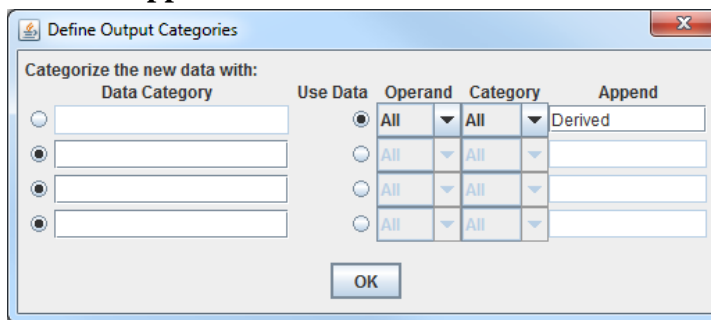
e. Click **Add Formula**.

6. Investigate the placement of the derived field in the *Field Selector* and move it under the *2D grid -> Derived* tree.

a. In the *Field Selector*, *Right-Click* on the **GFS CONUS 80km** data source and select **Reload Data**. Once this is done, the data source will be re-loaded and the new formula created earlier will generate a derived quantity. Note that this derived quantity is listed at the bottom of the **Fields** panel, and not under *2D*

grid -> *Derived* like the other 2D derived formulas (with remote data).

- b. Move this new derived field to under **2D grid -> derived** by editing the formula. To do this, *Right-Click* on **Formulas** in the **Field Selector** and select **Edit Formulas -> Derived Quantities -> MSLP – pressure**.
- c. In the **Derived** tab at the bottom of the **Formula Editor** window, click the **Define Output Categories** button.
- d. This **Define Output Categories** window allows for specifying where this derived field should be listed. To get this field to list under **2D grid -> Derived**, select *Use Data*, choose *All* for **Operand**, choose *All* for **Category**, then enter *Derived* for **Append**.

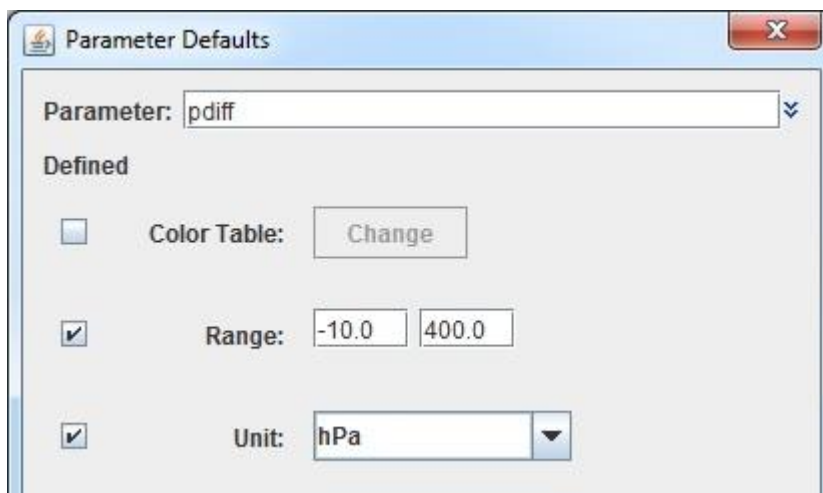


- The **Operand** dropdown refers to which operand in the formula (D1 and D2) will be used. This could be useful, for example, if the operands in the formula were from different categories such as a 2D and a 3D grid. Both of these grids are 2D so *All* can be used.
- The **Category** dropdown refers to which categories were defined when creating the formula. In this case, no categories were defined since specific field names were used. This could be used if an alias was used in the **Parameters** menu when creating the formula. For example, if TEMP (temperature) was specified as the alias, then there could be multiple categories (2D grid and 3D grid). This **Category** dropdown could then be used to say if you are putting the derived field under the 2D or 3D tree in the **Field Selector**.
- The **Append** menu informs McIDAS-V which menu tree (under the category tree) to place the derived field. In this case, the field is put under a tree called **Derived** which is under the **2D grid** tree (the tree where the MSLP and surface pressure fields are).

- e. Click **OK** to close the **Define Output Categories** window.
- f. Click **Change Formula** save the formula and close the **Formula Editor** window.
- g. Reload the data again using the method from 6a and now the derived field will list under **2D grid -> Derived**.

7. Display the data to see the difference between MSLP and surface pressure.


- a. In the *Field Selector*, choose the *2D grid -> Derived -> MSLP – pressure* field. Select the *Plan Views -> Contour Plan View* and the first 5 times. Click **Create Display**.
 - b. Probe the display with the middle mouse button to see the pressure difference in Pascals. Change the display to millibars by going to the *Layer Controls* for the layer and selecting *Edit -> Change Display Unit*. In this **Change Unit** window, use the dropdown to select *hPa*. Click **OK**.
8. It is possible to create a parameter default so millibars or hPa would be used as the display unit by when the display is created. This can be done by creating a parameter default.
- a. In the **Main Display**, select *Tools -> Parameters -> Defaults*.
 - b. From the *User Defaults* tab, select *File -> New Row*. In the **Parameter** menu, enter *pdiff*. This *pdiff* was defined in 5b ii as the name associated with the formula. Enter a **Range** of *-10 to 400* (or whatever range you think is suitable for the data). For **Unit**, use the dropdown to select *hPa*. Click **OK**.



- c. In the **Main Display**, create a new tab and then re-display the derived MSLP – pressure field. Note that the display unit and enhancement range specified in the parameter default are used to display the data.

Creating a Plugin

1. Remove all layers and data sources from the previous displays. Create a new, one-paneled map display and close any previously-existing displays.
2. Add a new NWS CWA map.
 - a. From the **Main Display** window, click on the blue *Default Background Maps* text in the **Legend** to get to the *Layer Controls* for the map lines.
 - b. From the *Layer Controls* tab of the **Data Explorer**, select *Edit -> Add Your Own Map*.
 - c. Click the **Browse** button next to **Map file or URL**.
 - d. Select the *<local path>/Data/Generic/MapLines/w_02oc18.shp* file and click **Open**.
 - e. For **Name**, enter *NWS CWA*.
 - f. Click **OK**.
3. Set McIDAS-V to default to drawing the NWS CWA maps in tabs made in the future.
 - a. In the *Layer Controls* tab of the **Data Explorer**, select *File -> Default Maps -> Save as the Default Map Set*.
 - b. Create a new, one paneled-map display in the **Main Display** window, and the NWS CWA map will be drawn.
4. Create a plugin that contains these map lines.
 - a. From the **Main Display Window**, select *Tools -> Plugins -> Create*.
 - b. From the **Plugin Creator** window, select *File -> Add File*.
 - c. From the **Open** window, select the *maps.xml* file and click **Open**.
 - d. From the **Plugin Creator** window, click the **Browse** button.
 - e. From the **Save** window, navigate to the directory where you want to write the plugin, and enter a name such as *nws_cwa_map_lines*. Click **Save**.
 - f. From the **Plugin Creator** window, click **Write Plugin**.
5. This plugin can now be passed around to other users. To install the plugin, the user would:
 - a. Select *Tools -> Plugins -> Manage* from the **Main Display**.

- b. From the **Plugin Manager**, select *File -> Install Plugin From File*.
 - c. Select the *nws_cwa_map_lines.jar* plugin and click **Open**.
 - d. Restart McIDAS-V.
6. For now, we will remove these CWA map lines so they no longer appear in all new display panels.
- a. From the **Layer Controls** of the map lines, click the *Remove The Map* () button next to **NWS CWA**.
 - b. Select *File -> Default Maps -> Save as the Default Map Set*.
 - c. Create a new tab, and the NWS CWA map lines will not be drawn.

Using NcML Aggregation

When working with local netCDF files that contain one timestep per file, to display a loop the easiest way is to merge the data from all of the files into one data source. Generally, this can be done through the **Gridded Data -> Local or General -> Files/Directories** chooser by selecting the *Aggregate Grids By Time* option in the **Data Type** dropdown menu. This data type uses the “time” dimension of each file to merge them together into one data source. However, not all files contain a “time” dimension, which means that *Aggregate Grids By Time* isn’t an option. For example, ABI netCDF files don’t contain a “time” dimension. In this case, NcML can be used to create a wrapper file that can specify how the time is defined for each file. Specifically, in this case, the date/time information is extracted from the file names.

1. Remove all layers and data sources, create a new one-paneled map display, and close any previously-existing tabs.
2. Load a single GOES-16 mesoscale-1 sector netCDF file as a gridded data source.
 - a. From the **Data Sources** tab of the **Data Explorer**, navigate to the **General -> Files/Directories** chooser.
 - b. Navigate to the `<local path>/Data/Satellite/ABI_data/Meso1` directory.
 - c. Select the first *OR_ABI_L1b* file in the listing.
 - d. Click **Add Source**.
3. Verify that no time dimension exists in the data.
 - a. From the **Field Selector** tab of the **Data Explorer**, *Right-Click* on the *OR_ABI-L1b* data source and choose **Properties**.
 - b. From the **Properties** window, open the **Metadata** tab.
 - c. This **Metadata** tab gives an “ncdump -h” output of the file, providing information about the structure of the data. Scroll to the top of this tab to see where the dimensions are defined and note that no “time” dimension is listed.
4. Load and display a loop of data from the NcML file.
 - a. From the **General -> Files/Directories** chooser, select the *meso_aggregate.ncml* file.
 - b. Click **Add Source**.
 - c. From the **Fields** panel of the **Field Selector**, select **2D grid -> ABI L1b Radiances**.
 - d. From the **Displays** panel, select **Imagery -> Image Display**.

- e. Note that the *Times* tab lists 10 times, which matches the number of netCDF files in the data directory. Click **Add Source**.

5. Inspect the display in the **Main Display** window. The data is displayed at the native resolution of GOES-16 band 2 ABI data (0.5km). The Time Animation Widget can be used to play through the loop.

The text of this NcML file is:

```
<netcdf xmlns="http://www.unidata.ucar.edu/namespaces/netcdf/ncml-2.2">
  <aggregation dimName="time" type="joinNew">
    <variableAgg name="Rad"/>
    <variableAgg name="DQF"/>
    <remove name="t" type="variable"/>
    <scan location="" dateFormatMark="OR_ABI-L1b-RadM1-M3C02_G16_s#yyyydddHHmmss" suffix=".nc"
subdirs="false"/>
  </aggregation>
</netcdf>
```

The key part in this file is the “scan location” line, where dateFormatMark is used to specify which characters in the filename.

Additional information about NcML aggregation can be found here:

<https://www.unidata.ucar.edu/software/netcdf-java/current/ncml/Aggregation.html>

Note that NcML can also be used to fix netCDF files that aren't CF-Compliant. If a user tries to load a non-compliant netCDF file, a Non-Compliant NetCDF Widget tool will appear. Once this happens, the user can run their file through an online CF-Compliance checker, such as:

<https://pumatest.nerc.ac.uk/cgi-bin/cf-checker.pl>

Once it has been determined what is wrong with the file, NcML can be written to make corrections to the file. This could include removing or editing variables or attributes (variable and global). A good resource giving examples of how this can be done is here:

<https://www.unidata.ucar.edu/software/netcdf-java/current/ncml/Cookbook.html>

Scripting Introduction

Python vs. Jython

Fact: You will do all of your McIDAS-V programming in the Python programming language.

Fact: McIDAS-V uses an implementation of the Python programming language called Jython.

Fact: The original and most widely used implementation of the Python programming language is not Jython; the full and proper name for that one is actually CPython. (In case you're wondering, Jython is implemented in Java, and CPython is implemented in C!). In day-to-day usage, CPython is often referred to as just "Python."

What does all that mean?

As you learn McIDAS-V scripting, you should know that all the documentation you'll find across the web for the Python *programming language* (specifically, version 2.7) is relevant and accurate. Jython is very careful to retain compatibility with the Python language. This includes almost the entire Python standard library, which is why we have access to the functions we need to import, like **glob** and **basename**.

However, the most important distinction between Jython and CPython is the availability of libraries. Because Jython is Java-based, we do **not** generally get to use the Python libraries that depend on "native" code. As a result, there is a large list of libraries we **cannot** use in Jython, including:

- Scipy
- NumPy
- matplotlib
- netcdf4-python
- h5py

Once you are comfortable with the basics of the Python programming language, the remainder of learning McIDAS-V scripting is largely about learning the "McIDAS-V way of doing things": instead of NumPy, matplotlib, and netcdf4-python, we will use the VisAD data model, VisAD displays, and NetCDF-Java to get our work done.

In summary, Jython's language and standard library are the same as CPython, but many non-standard Python libraries such as NumPy are not available in Jython. However, McIDAS-V has good alternatives for data access, plotting, and numerical computation.

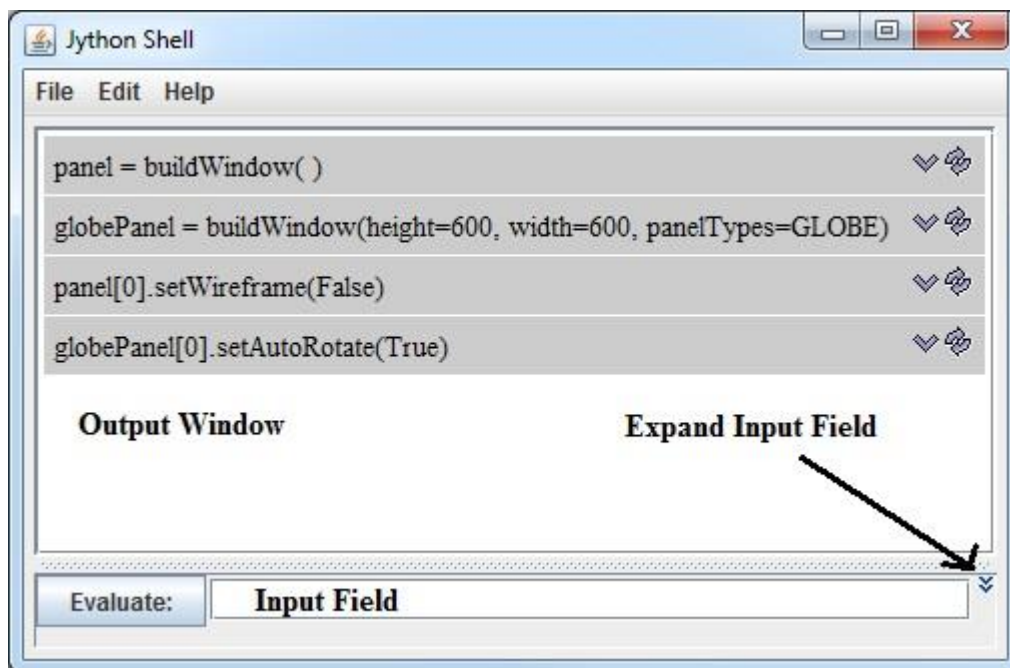
For more information about McIDAS-V scripting functions, please see the User's Guide and the McIDAS-V scripting and advanced scripting tutorials available online.

Jython Shell Scripting

The **Jython Shell** consists of an *output window* on top and an *input field* on the bottom. The user enters Jython into the *input field*. When the Enter key or "**Evaluate**" is pressed, the Jython input is evaluated and output is shown in the *output window*. The **Jython Shell** is a great tool to begin writing scripts that can be run from the background. When inputting commands, the **Jython Shell** runs in single or multi-line mode. You can switch modes by using the double down arrows or with the shortcut **Ctrl+/,** The **Evaluate** button also has a shortcut of **Shift+Enter.**

Here is a chart containing keyboard shortcuts for the **Jython Shell**:

Enter	Evaluate command in single-line input mode
Shift+Enter	Evaluate command in single or multi-line input mode
Ctrl+/,	Switch between single and multi-line input modes
Ctrl+p	Recall a previously evaluated command
Ctrl+n	Recall the next run command, assuming Ctrl+p was already used



1. Using the **Jython Shell**, create a window with a single panel **Map Display**.
 - a. From the **Main Display** window, select **Tools -> Formulas -> Jython Shell** to open the **Jython Shell**.
 - b. In the *input field* of the **Jython Shell**, type:


```
panel = buildWindow()
```
 - c. Click **Evaluate**.

buildWindow is the function used to create an object that contains an array of panels. This created a window, just as you would through the GUI with **File -> New Display Window -> Map Display -> One**

Panel.

2. Now, create another window, this time with a globe display.

a. In the *input field* of the **Jython Shell**, type:

```
globePanel = buildWindow(height=600, width=600, panelTypes=GLOBE)
```

b. Click **Evaluate**.

Now, there are two single-paneled displays, each of which can be modified.

3. Turn off the wireframe box in the map display window.

a. In the *input field* of the **Jython Shell**, type:

```
panel[0].setWireframe(False)
```

b. Click **Evaluate**.

The “[0]” is needed because each window contains a list of panels, and “0” operates on the first one.

4. Enable auto-rotation in the globe display window.

a. In the *input field* of the **Jython Shell**, type:

```
globePanel[0].setAutoRotate(True)
```

b. Click **Evaluate**.

5. Close the two display windows created in steps 1 and 2 above.

Basic Jython Terminology

Three important terms are: *function*, *method* and *object*. In the most general terms, an object is returned from a function and a method operates on an object and may return a new object.

In steps 1 and 2, the **buildWindow** function was used to create an object, in this case an array of panels. Objects can have one or more attributes and these attributes are defined by a class. Methods are used to operate on an object. In step 3, **setWireframe** operated on the panel object by turning off the wireframe box.

The word *object* can be intimidating because it hints at the topic of *object-oriented programming*, which can be complex and confusing. However, as McIDAS-V programmers, we just need to know that an object is a special kind of variable that has *functions* associated with it. These special kinds of *functions* are referred to as *methods*.

Display a Remote GOES-16 Image

1. From the **Jython Shell**, click the **Expand Input Field** button to allow for multi-line input mode.
2. Create a display of GOES-16 mesoscale-1 sector band 13 temperature data.

- a. In the *input field* of the **Jython Shell**, type:

```
panel = buildWindow()

addeParms = dict(
    server = 'lead.unidata.ucar.edu',
    dataset = 'RTGOESR',
    descriptor = 'M1C13',
    unit = 'TEMP',
    size = 'ALL',
)

data = loadADDEImage(**addeParms)

layer = panel[0].createLayer('Image Display', data)
```

- b. Click **Evaluate**.

As an alternative to typing the above, you can open the *<local path>/Data/Scripts/McIDAS-V/image_display.py* script in a text editor, and copy/paste its contents into the *input field*. Note the 4-space indentation is necessary in the `addeParms` dictionary.

- c. Description of the above script:

- `panel = buildWindow`
This function builds a window, as was demonstrated previously
- `addeParms`
This is a Python dictionary that contains keys and values matching keywords that the `loadADDEImage` function accepts. These keywords are used to define the server where the data is being pulled from, along with the dataset and descriptor names. The display unit and size of the image are also defined here.
- `data= loadADDEImage`
This function creates a data object using the keys/values defined in the `addeParms` dictionary.
- `layer=`
This function adds a data layer to the display window created at the beginning of the script

- Note that the documentation of the McIDAS-V scripting functions (buildWindow, loadADDEImage, createLayer, and subsequent commands) can be found in the McIDAS-V User’s Guide.

3. Determine statistical information about the data.

- a. In the *input field* of the **Jython Shell**, type:

```
print describe(data)
```

- b. Click **Evaluate**.

- c. Description of the above command:

- The McIDAS-V function, describe, accepts a data object. Here, we are passing through “data”, which is the data object that was returned from loadADDEImage in the script run previously.

4. Change the enhancement of the displayed layer.

- a. In the *input field* of the **Jython Shell**, type:

```
layer.setEnhancement('ABI IR Temperature')
```

- b. Click **Evaluate**.

- c. Description of the above command:

- The McIDAS-V function, setEnhancement, applies an enhancement to the layer object created by createLayer from the script run previously. This command accepts a string of an enhancement name.

5. Capture an image of the display.

- a. In the *input field* of the **Jython Shell**, type:

```
panel[0].captureImage('<local-path>/JythonShellTest.jpg')
```

- b. Note that you should substitute “<local-path>” with a directory location on your machine that you have access to. Click **Evaluate**.

- c. Description of the above command:

- The McIDAS-V function, captureImage, captures an image of the display panel returned from the buildWindow function from the script run previously.

- d. Outside of McIDAS-V, use a software, such as a browser, to view the image captured.

Background Scripting

1. Exit McIDAS-V.
2. Using a text editor, edit the `<local path>/Data/Scripting/McIDAS-V/sandwich_example.py` script. The last line in the script is the `captureImage` command. In this command, change “`local_path`” to a directory location on your machine that you have write access to. Save the script.
3. From a Terminal or Command Prompt window, `cd` to your McIDAS-V-System, which should be located in the directory where McIDAS-V was installed.
4. From the Terminal or Command window, run the script using the “-script” flag. Note that you should substitute “`local_path`” with the location of the script on your machine. Run the command below matching your operating system:

On Windows:

```
runMcV.bat -script /local_path/sandwich_example.py
```

On Linux:

```
runMcV -script /local_path/sandwich_example.py
```

On macOS:

```
./runMcV -script /local_path/sandwich_example.py
```

5. Once the script has completed, use a software, such as an internet browser, to view the captured image.
6. Description of the script:
 - a. `addeParms`:
Dictionary to pass through `loadADDEImage`. Two bands are loaded, and the keys/values in this dictionary apply to both bands.
 - b. `b3Data/b13Data=loadADDEImage`:
`loadADDEImage` commands for band 3 and band 13 data. A few keywords specific to each band are specified, along with the `addeParms` dictionary
 - c. `rgbData=sandwich`:
Runs the `sandwich` function to generate a sandwich data source from the band 3 and band 13 data objects
 - d. `visLayer/rgbLayer`:
Individually display the visible and RGB (`sandwich`) data objects. The RGB layer is then updated to lower the gamma value, making it brighter.
 - e. `panel[0].captureImage`:
Captures an image of the display window.