User Guide for Tycho v13

By Daniel Parrott

Last updated April 23, 2026

Contents

Introduction

Welcome to Tycho. This software is designed to facilitate the detection and measurement of asteroids, comets, variable stars, and artificial satellites. It also supports a technique known as synthetic tracking, which enables the detection of very faint moving objects. This is achieved by using the data from many exposures (a minimum of 11) and applying the stacking technique for an improved signal-to-noise ratio. Because the exposures are stacked in thousands of different possible combinations, synthetic tracking allows for the detection of faint objects even when the motion of the object is unknown. Since 2021, over 100 Near Earth Objects (NEOs) and more than five comets have been discovered by amateur astronomers using Tycho.

Tycho also makes it easy to construct lightcurves from a series of input images. It features a Period Search module to determine the period from such a lightcurve, and can work with both stationary sources (variable stars, exoplanets) as well as moving sources (asteroids and more).

Supported Operating Systems

- macOS 11.0 or later
- Windows 7 SP1 or later

Terminology

Term/Acronym	**Definition**
CPU	Central Processing Unit Your machine has one or more CPUs. A CPU is designed to carry out instructions in a procedural fashion (one instruction at a time). This is ideal for database transactions, spreadsheet functions, and so forth.
GPU	Graphics Processing Unit Your machine may have one or more GPUs. A GPU excels at performing a single instruction in a massively parallel fashion (applying the same instruction across a large amount of data).
Dedicated GPU	A dedicated GPU is one that has its own source of video memory. Such cards are typically much more powerful than their “integrated” counterparts.
Integrated GPU	An integrated GPU does not have its own memory. Instead, it uses the available memory from the system, and is typically less powerful than a dedicated GPU.
OpenCL	Open Compute Language One of the languages that can interface with a graphics processor is called OpenCL. This is a cross-platform language, meaning that it can work with AMD, NVIDIA, and Apple Silicon.
Metal	A proprietary interface developed by Apple for GPU acceleration on macOS systems. The macOS version of Tycho supports both OpenCL and Metal, with Metal typically outperforming OpenCL on the newer Apple Silicon systems.

First Time Setup

Enable GPU Acceleration (if available)

By default, Tycho will be installed to operate in "CPU mode", with no GPU acceleration. If your system has the requisite hardware, then it is recommended to enable GPU acceleration. This is accomplished by going to *Settings->GPU Acceleration* from the main menu. Figure 1 shows an example configuration.

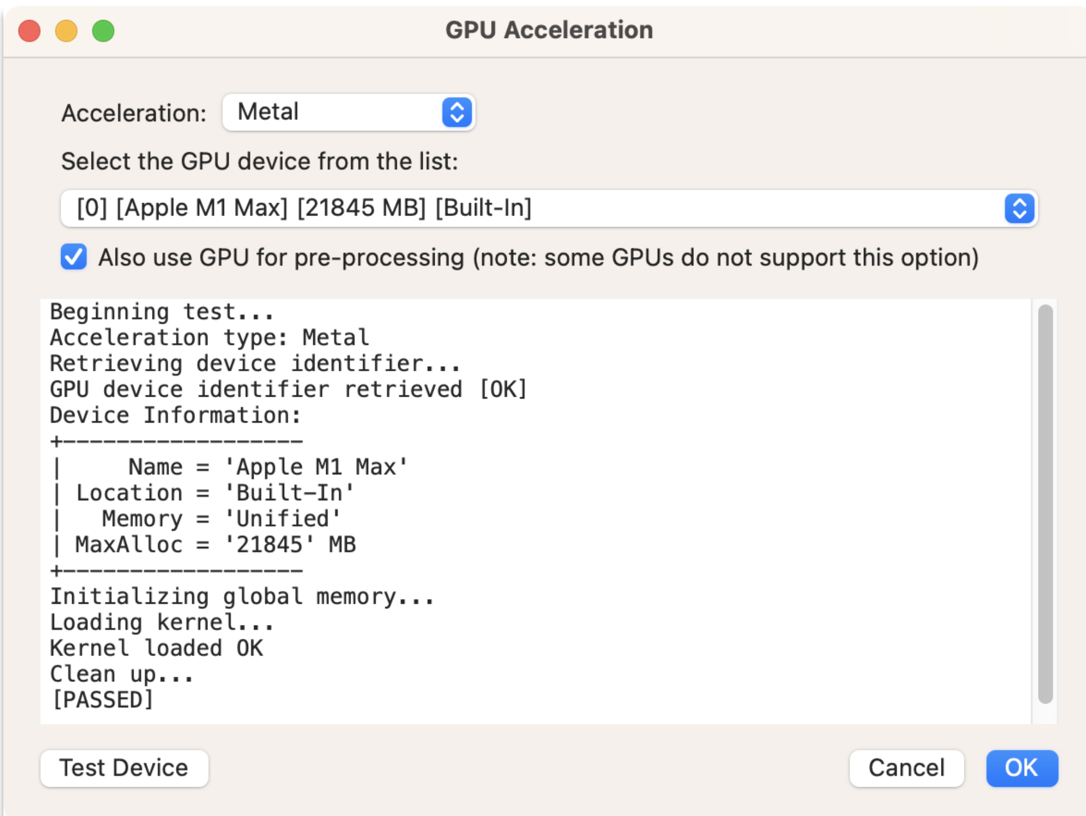


Figure 1 – GPU Acceleration Settings

Troubleshooting (for Windows operating system)

1) If an error occurs when choosing a GPU device, ensure that the latest drivers are installed for the graphics card. If after updating drivers you continue to get an error, try renaming the provided "OpenCL.dll" file that comes with Tycho to "OpenCL.dll_old" so that Tycho will use the OpenCL library provided by the system. The provided DLL file is located in C:\Program Files\Tycho
2) If the program unexpectedly quits when checking the box to enable GPU acceleration, it is likely due to having Intel UHD 630 graphics drivers installed alongside the dedicated NVIDIA or AMD drivers. To remedy this, you can go to Device Manager and disable the Intel graphics driver. This can be done by opening Control Panel, choosing Device Manager, and under the "Display Adapters" section, right-click on the Intel UHD 630 device and choose "Properties". Then, on the "Driver" tab, click the "Disable" button.

Configure Report Parameters

The report parameters are used to specify the persons involved in the acquisition and measurement of the asteroid image data.

Navigate to **Settings->Report Parameters** and adjust the settings as needed:

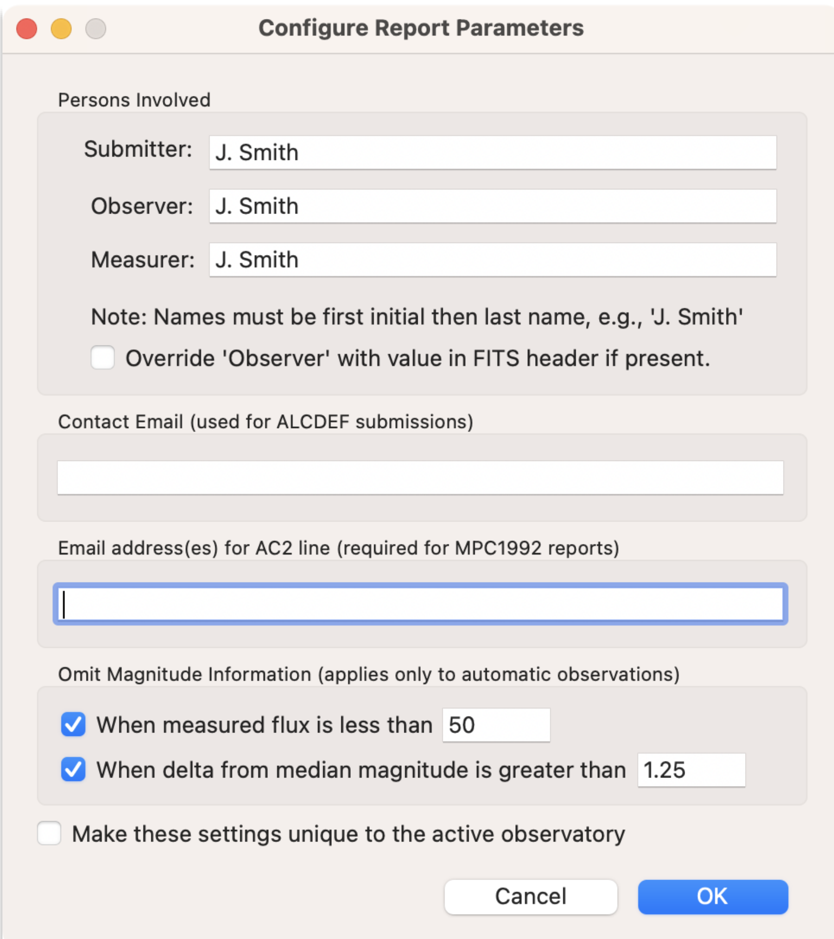


Figure 2 - Report Parameters

Submitter: Specify the name of the person submitting the reports.

Observer: Specify the name of the person who acquired the images.

Measurer: Specify the name of the person who processed and generated the astrometric and photometric measurements.

Override: Check this box if you want to replace the 'observer' field with a value supplied in the FITS header.

Contact Email: Specify a contact email for ALCDEF (lightcurve) submissions.

Email address(es) for AC2 line: You can specify notification email addresses for MPC submissions. Use a comma to separate multiple addresses.

Omit Magnitude Information: These setting apply only to observations generated via "Verify Track".

Typically, if you wish to share discovery credit with another person, you would designate one person as the "Observer", and the other person as the "Measurer".

All names should be formatted as first initial followed by last name, e.g., "J. Smith".

"Make these settings unique to the active observatory": Check this box if you wish to have unique report parameters for the active observatory. For example, you might use Tycho for multiple observatories, and if you want them to use different reports, you can check this box. Otherwise, if unchecked, they will all share a common set of report parameters.

Configure Observatory Information

In order to generate reports that can be submitted to the Minor Planet Center (MPC), it is necessary to provide some information about the observatory that captured the images. Navigate to *Settings->Observatory* from the main menu. Then choose *Action->Add Observatory*, as shown in Figure 3.

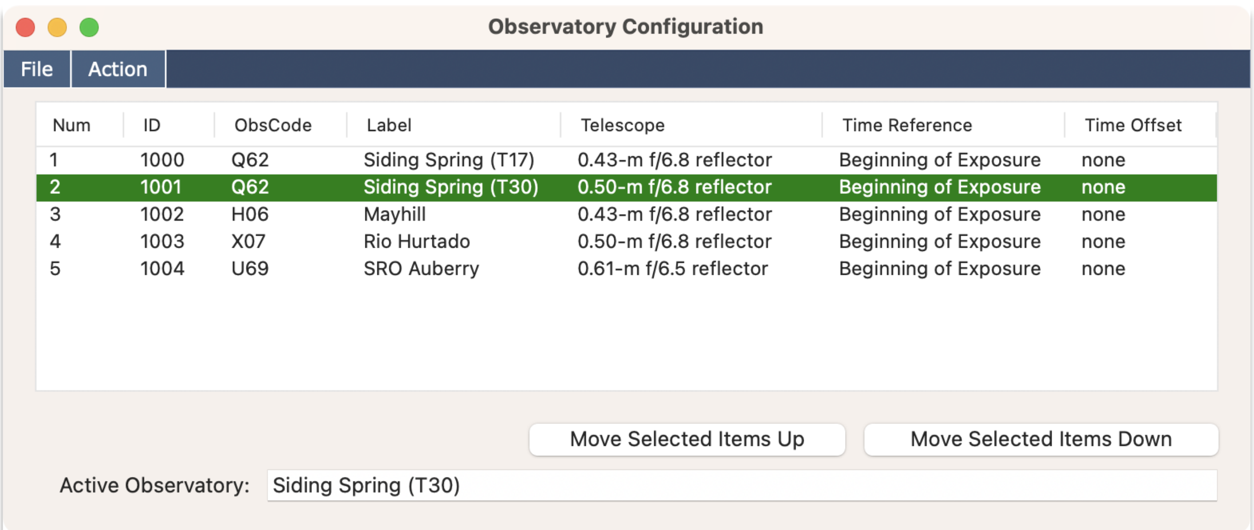


Figure 3 - Observatory Configuration

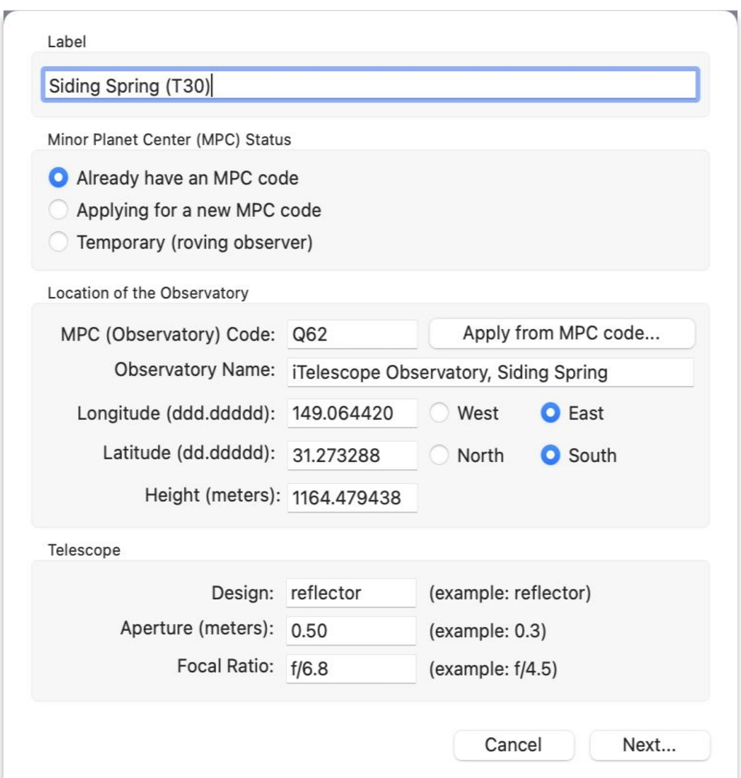


Figure 4 - Add New Observatory

Label: Give the observatory a label so that you can refer to it later.

Observatory Status: If the observatory already has a code from the MPC, choose “Already have an MPC code”. If the observatory is applying for a new MPC code, choose “Applying for a new MPC code”. If the observatory is temporary (such as a mobile location), choose “Temporary”.

Location of the Observatory: Populate these fields as appropriate. If the observatory has an MPC code, you can click “Apply location from MPC code” to populate the fields automatically. You may need to update the “ObsCodes” file (Settings->ObsCodes).

Telescope: Specify the type and aperture of the telescope, as well as its focal ratio.

Specify Settings for Camera

Click “Next…” to proceed to the camera details for the new observatory.

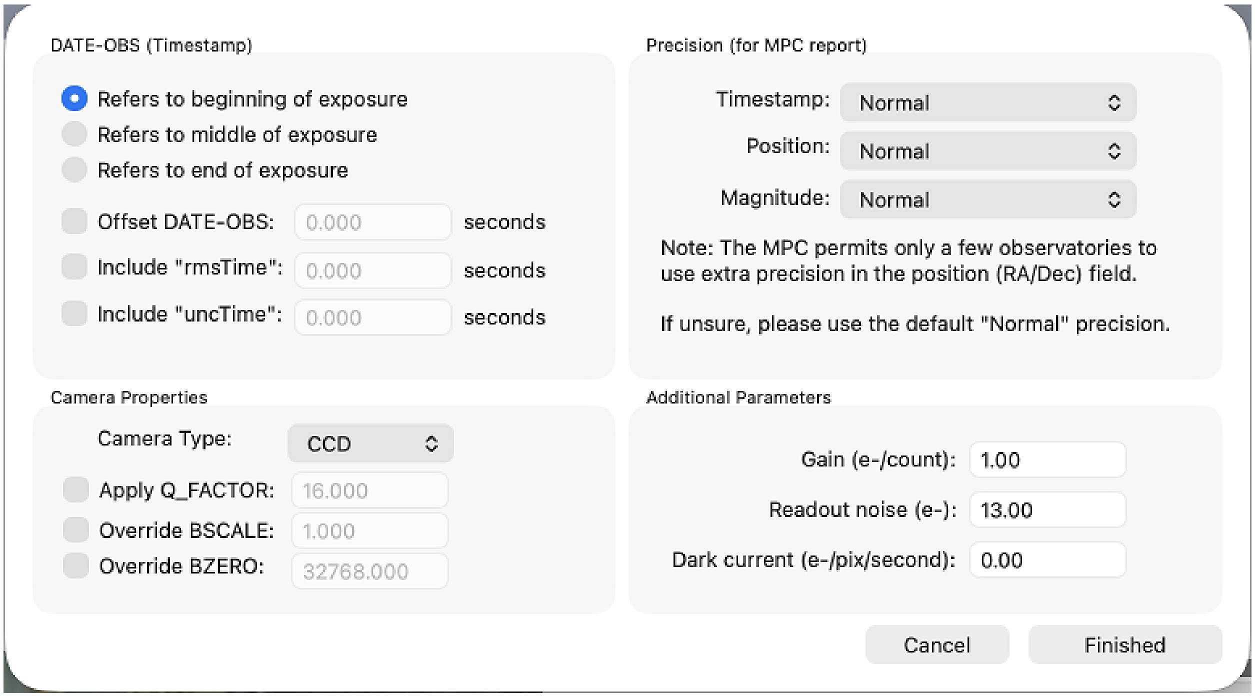


Figure 5 - Camera Details

DATE-OBS: In order to make accurate observations, it is critical that you specify the correct setting for the time information provided by the camera. Most cameras typically generate a “DATE-OBS” keyword in the FITS header of the image that refers to the beginning of the exposure. However, some newer CMOS cameras may specify a “DATE-OBS” timestamp that refers to the *end* of exposure.

Offset: If you know that the DATE-OBS timestamp is off by a fixed amount, you can specify a time offset here. However, it is preferred that you fix the clock to be accurate in the first place, if possible.
rmsTime: Random uncertainty in observation time. Used only for ADES reports. Optional.
uncTime: Estimated systematic time error. Used only for ADES reports. Optional.

Precision: For very fast-moving objects (satellites, very close NEOs), it may be necessary to use extra precision in the timestamp field. Note: the MPC does not allow most observatories to use extra precision in the position (RA/Dec) field.

Camera Properties: Choose the camera type, either CCD or CMOS, from the drop-down.
Apply Q_FACTOR: If the ADC of the camera is more than 16-bits, you can specify a quantization factor to compress the image data into 16-bit space.

Additional Parameters: These parameters are used for SNR calculation.

Finally, click “Finished” and the new observatory will appear in the list of observatories. To make the observatory the active observatory, right-click on it and choose “Make Active” from the pop-up menu that appears.

If you ever need to modify the observatory information, simply right-click the observatory in the list and choose “Edit…” from the pop-up menu that appears.

If there are multiple cameras in use by the same observatory, with different time settings, then you can simply add additional observatories to the list with the same location data but different time settings. Assign these observatories different labels so as to distinguish between them internally.

In essence, an “observatory” in this context refers to a set of configuration values that can be applied to a set of images. As a result, the “Observatory Configuration” window allows you to easily switch between different configurations by right-clicking on the desired configuration and choosing “Make Active”.

Configure the Find_Orb Software

The Find_Orb software, written by Bill Gray, can be used to compute orbits from a set of measurements. By default, Tycho can use an online variant of this software, however it is also possible to configure a local variant as well. To do so, navigate to ***Settings->Find_Orb*** from the main menu and choose the desired configuration: "online service" or "local installation".

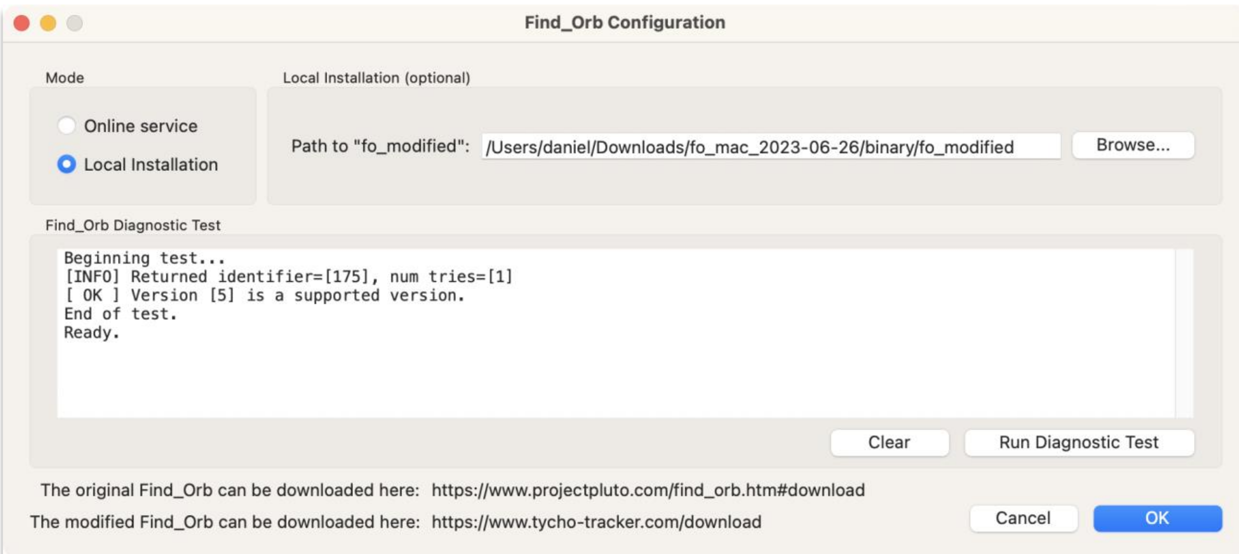


Figure 6 - Configuring the Find_Orb Software

The local variant, which has been modified for Tycho, can be downloaded here:

https://www.tycho-tracker.com/download

An example configuration is shown in Figure 6.

Note for macOS users: If you have placed the Find_Orb binary inside the Downloads or Desktop folder, be sure to grant Tycho access to the Downloads and/or Desktop folder.

When finished, click "OK" to save the settings.

Configure the Known Object Database

The "Known Object" database is useful to determine which detections match up with already known objects. It can be configured by going to *Settings->Known Objects* from the main menu.

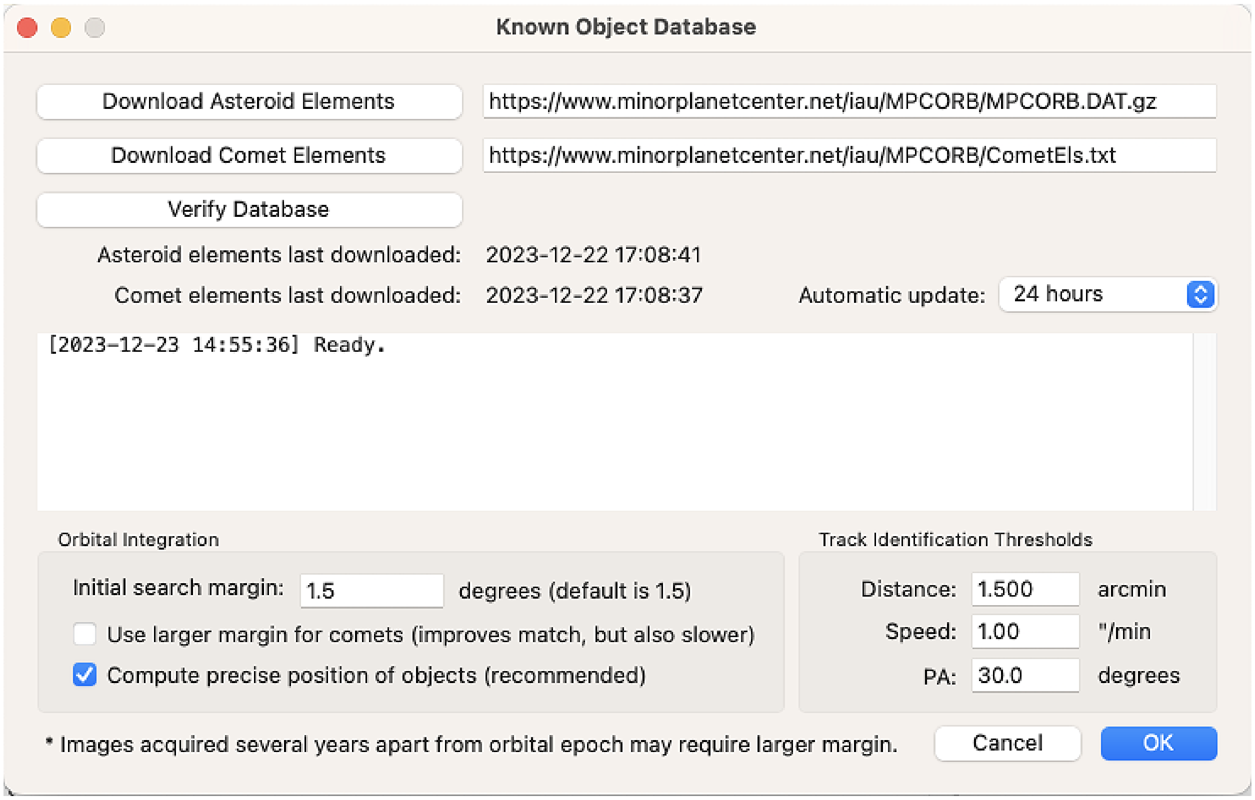


Figure 7 - Known Object Database Configuration

In the top-left there is a button "Download Asteroid Elements". Click this button to download the orbital elements of over 1 million asteroids. When it has completed, proceed to click the button "Download Comet Elements". Finally, when that has completed, click "Verify Database". There should be over 1 million asteroids and over 900 comets listed.

Configure the other settings as shown in Figure 7. If necessary, you can adjust the search margin parameters for object identification, but usually the default parameters are optimal.

Configure the Star Catalog

Proceed to ***Settings->Star Catalog*** to configure the appropriate settings. For most photometry work, it is recommended to use the new ATLAS catalog (also known as ATLAS-REFCAT2). The Downloads page on the Tycho website includes three different links to this catalog, allowing you to choose the desired coverage from magnitude 16 up to magnitude 20. Refer to the notes on the ATLAS tab for details on how to acknowledge use of this star catalog in a published paper.

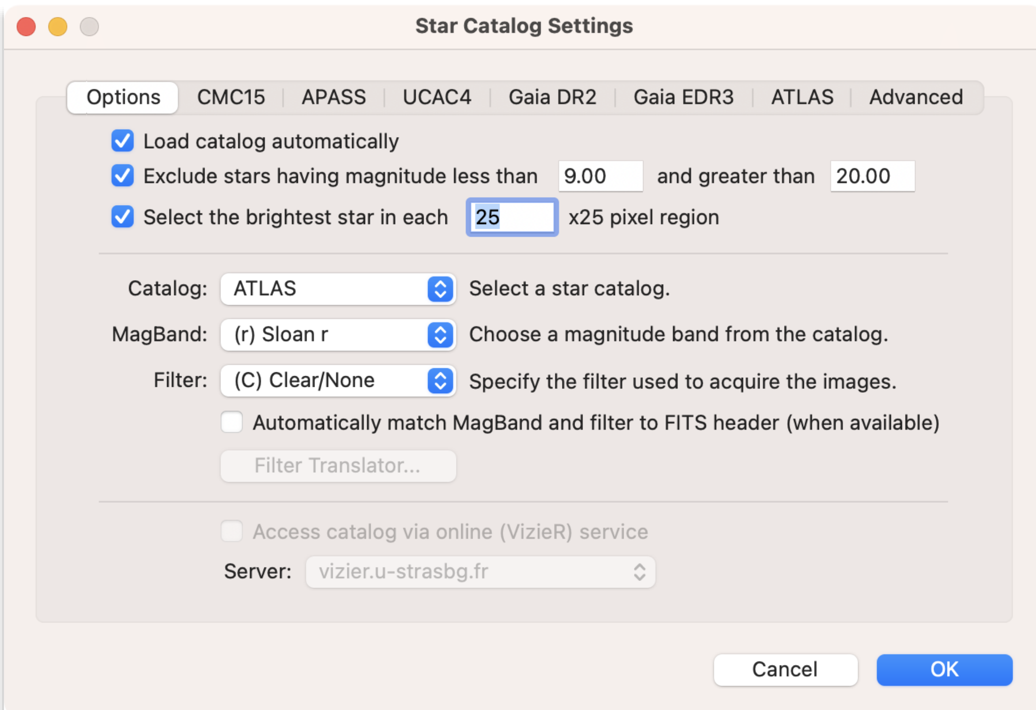


Figure 8 - Configuring the Star Catalog

To configure an offline star catalog (such as ATLAS), navigate to the relevant tab and specify the full path to the folder that contains the "de_*" folders ("z*" in the case of UCAC4). Then, navigate back to the "Options" tab and select the desired catalog from the dropdown menu next to "Catalog".

Primary Use Case	**Recommended Star Catalog**	**MagBand**	**Filter**
Astrometry of asteroids	Gaia EDR3	(G) Gaia	None
Photometry of asteroids	ATLAS	(r) Sloan R	None
Photometry of variable stars	ATLAS if wanting ensemble. AAVSO otherwise.	Should match the filter	B, V, R, I

For asteroids, it is recommended not to use a filter, so that you can capture as much light as possible on these faint targets. For photometry of variable stars, the ATLAS catalog can be used if wanting an ensemble of comparison stars. However, it is also possible to use AAVSO comparison stars regardless of the chosen catalog. This is achieved by overriding the catalog from within the "Image Viewer" photometry menu, "Download AAVSO Chart". More details are provided in the photometry section.

To summarize: If in doubt, use the ATLAS catalog. It can be overridden by AAVSO comp stars later on.

Configure the Alignment Settings

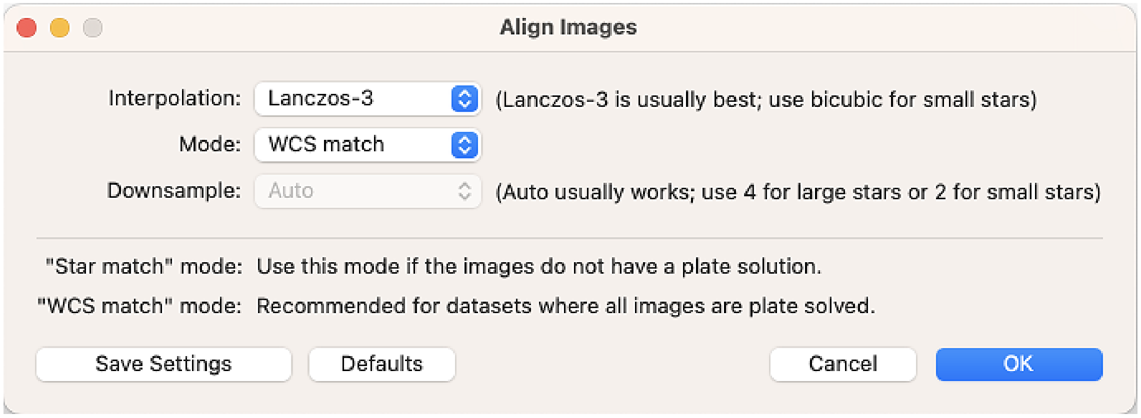


Figure 9 - Alignment Settings

Navigate to ***Action->Align Images*** to specify the settings for the alignment routine. Refer to Figure 9 for an example configuration.

Configure Observatory Codes

If you have not already done so, you may want to configure the Observatory Code file, “ObsCodes.html”. This file allows Tycho and *Find_Orb* to recognize the different observatory codes that have been issued by the Minor Planet Center.

You may update the file by going to **Settings->ObsCodes** and clicking the Download button. Then, if you have configured a local copy of *Find_Orb*, you can also keep it updated by clicking the button labeled “Copy to Find_Orb”. Note: if you are using Find_Orb via the “Online Service” interface, there is no need to copy the observatory codes as the online service automatically stays up-to-date.

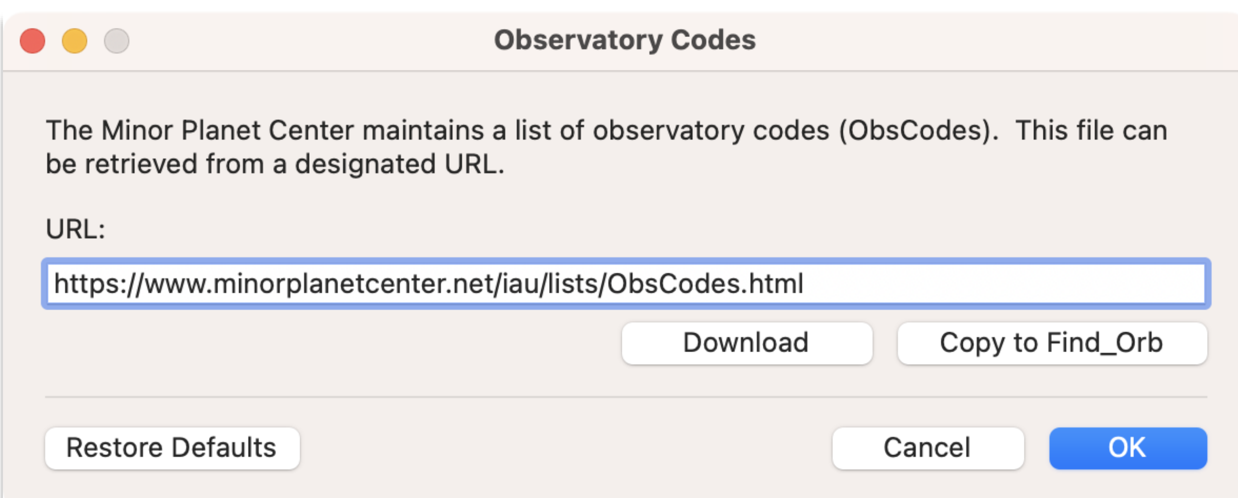


Figure 10 - Loading Observatory Codes

Configure the Plate Solver

Plate solving is necessary an order to determine image characteristics such as plate scale, orientation, and sky coordinates. Usually only the first (or reference) image needs to be solved, since the next step – alignment – will copy the plate solution of the reference image to all other images.

Tycho considers an image to be "plate solved" when two conditions are satisfied. The first is that the image must contain World Coordinate System (WCS) entries inside its FITS header. The second condition is that the entries must correspond to a 3rd-order SIP polynomial.

There are two ways to plate solve an image. The first of which is to use the Tycho online service which invokes the `solve-field` software developed by Dustin Lang et al. This is the easiest and preferred technique for most users, and is also very fast because it does not upload the image data – only the extracted sources. The second option is to use an external solver, being sure to specify a 3rd order solution.

Plate Solve Option 1: Using the Tycho Online Service

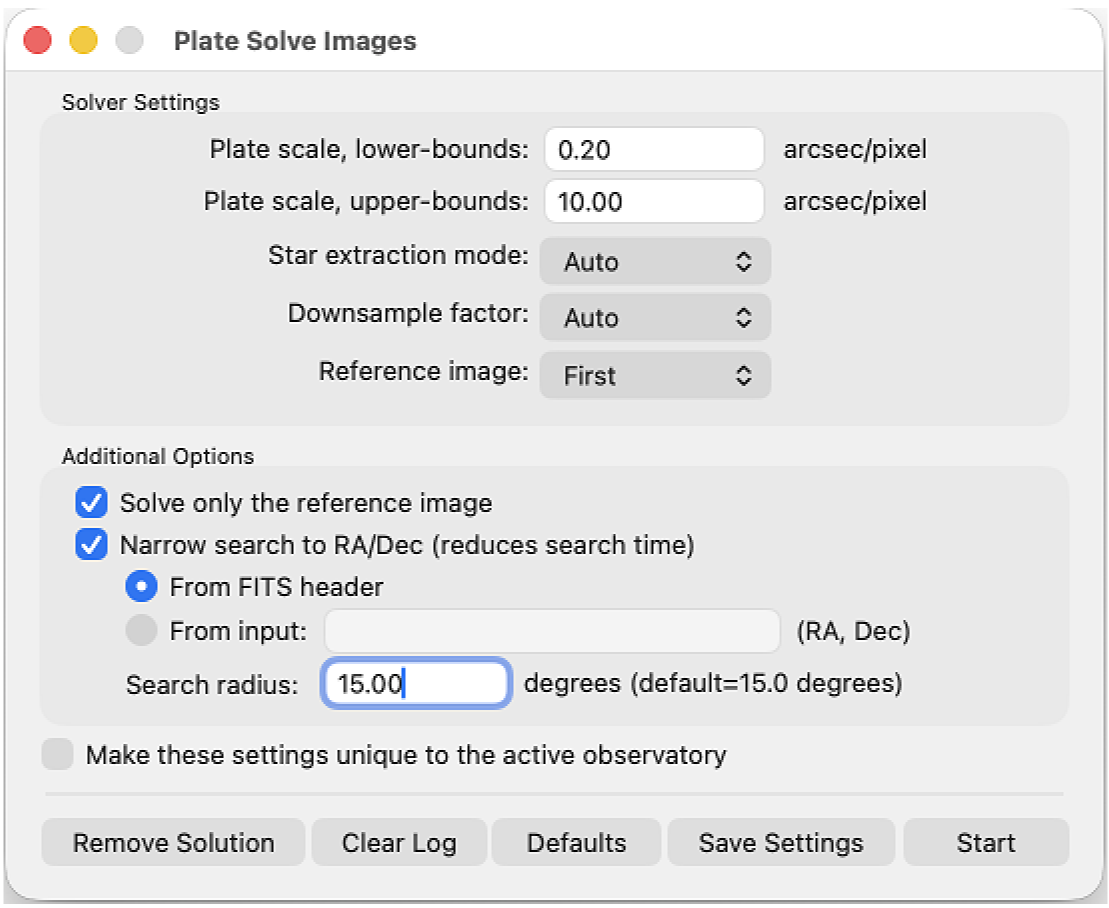


Figure 11 - Settings for Tycho Online Solver

As mentioned, the Tycho online service is recommended for most users as it is quick, easy, and reliable. Navigate to ***Action->Plate Solve Images*** and configure the solver as shown in Figure 11. Then click "Save Settings" to save the settings for future use. When you are ready to plate solve, click the "Start" button.

Plate Scale: If you know the plate scale of your imaging chip, you can provide a lower- and upper-bound on the plate scale. For example, if your plate scale is 1.6, you may specify 1.4 as the lower bound and 1.8 as the upper bound. If you are unsure, it is perfectly fine to specify a wider range such as 0.5 to 3.0.

Star Extraction: Auto, standard, or extended. Auto usually works and is recommended. Standard would apply to images having a reasonable number of stars while extended mode can be useful for images having few stars. However, extended mode is more sensitive to camera artifacts like hot pixels.

Downsample: Auto mode usually works and is recommended as the default. The higher downsample factors would apply to large images, while if you have smaller images you would try a lower downsample factor. Example: one might use downsample of 4 for a 6000x6000 sized image, while using downsample of 1 for a 700x700 sized image.

Reference Image: "First" usually works and is recommended as the default. Alternatively, you may choose auto, middle, or last as the reference image. **Note**: The "Plate Solved" indicator on the Image Manager will not show "Yes" until the first image has been solved. In other words, the plate solve can be successful and still indicate "No" because the reference image was set to other than "First". However, once alignment is performed, the plate solution of the reference image will be copied to all other images in the dataset -- including the first image -- and the indicator will then show "Yes".

Narrow Search: This option is usually recommended as it can dramatically speed up the search, as well as prevent false matches from occurring. Note however that some FITS images may contain incorrect RA/Dec values (off by more than the 15.0 degree search radius) and therefore lead to a failed solution. In such a (rare) scenario, one should leave this option unchecked.

Plate Solve Option 2: Using an External Solver

As an example of how to use an external solver, here are the steps to solve an image using the astrometry.net online service. Note again that you need only solve the first image in the dataset.

First, navigate to http://nova.astrometry.net and click on the "Upload" menu item.

Before uploading the file, click on the "Advanced Settings" text and scroll down to the following settings.

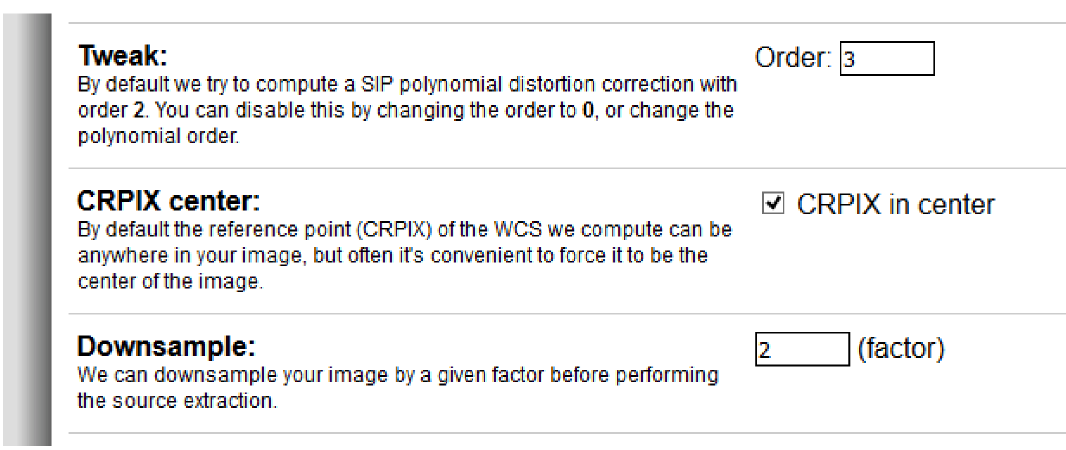


Figure 12 - Advanced Settings for Online Plate Solver

“Tweak” should be set to 3 in order to satisfy the 3rd-order polynomial requirement.
“CRPIX center” should ideally be checked, but it is not usually problematic if left unchecked.
“Downsample” is by default 2, but you may find that 4 works better for some images.

Once these Advanced Settings have been specified, proceed to upload the image and wait for the service to return with a result. When the service has finished, you can download the plate solved image by clicking on the “new-image.fits” link, as shown in Figure 13.

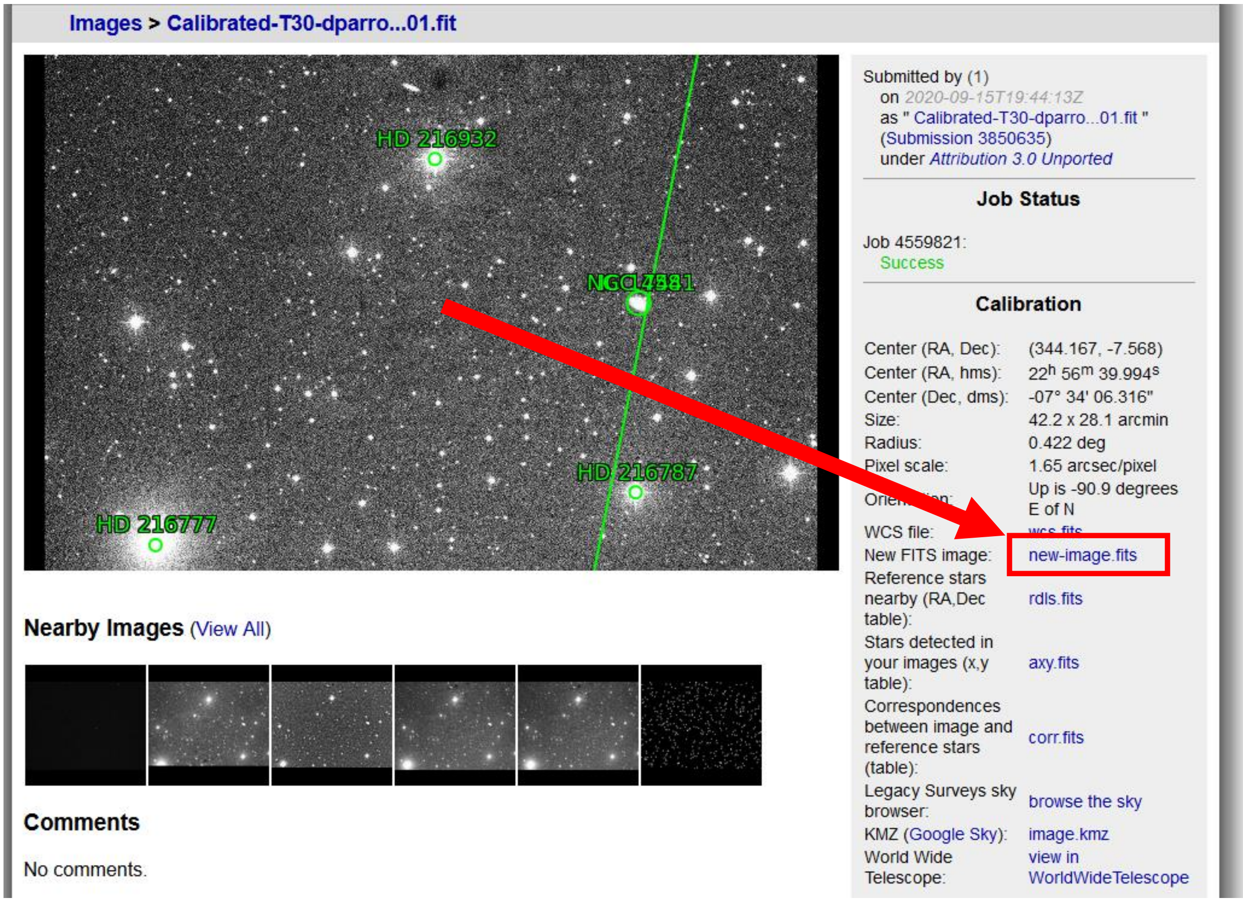


Figure 13 - Retrieving the Plate Solved Image

Once downloaded, you can replace the unsolved first image with “new-image.fits”, which will take the place of the first image in the image sequence. As you can see, using an external solver is a bit more tedious than using the integrated solver, but it provides one more option available should it be desired.

Acquiring an Observatory Code

The Minor Planet Center (MPC) is the official authority behind the process of requesting and acquiring an observatory code.

This page from the MPC website provides some details on acquiring an observatory code:
https://www.minorplanetcenter.net/iau/info/Astrometry.html

Specifically, this section on the page provides the pertinent details:
https://www.minorplanetcenter.net/iau/info/Astrometry.html#HowObsCode

Until you have a code, you will want to make sure that the option "Applying for a new MPC code" is selected as the "Observatory Status" (refer to the section "Configure Observatory Information").

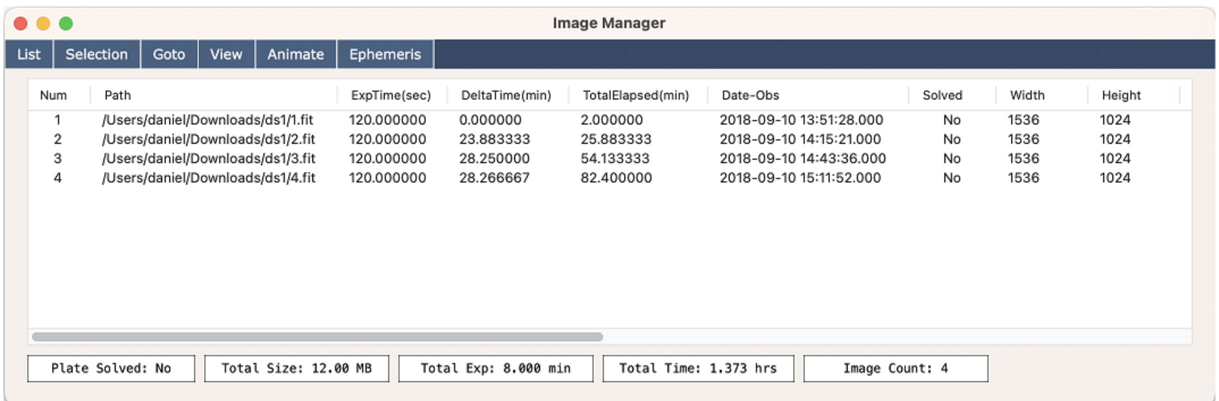
Note: The example dataset provided in this tutorial was taken from an observatory that already has a code (Q62). When applying for a new code, you must use images taken from your own observatory.

Example #1: Measuring a Known Asteroid (using only 4 images)

The purpose of this first example is to demonstrate how to use Tycho with a minimum set of images for basic measurement. Later examples will show how to use the synthetic tracker, which requires at least 11 images, enabling the detection of very faint asteroids.

Step 1: Load the Images

Launch Tycho. From the “Image Manager”, choose ***List->Add Images***, and load the images contained in the example dataset labeled “ds1”. There should be exactly four images.



Num	Path	ExpTime(sec)	DeltaTime(min)	TotalElapsed(min)	Date-Obs	Solved	Width	Height
1	/Users/daniel/Downloads/ds1/1.fit	120.000000	0.000000	2.000000	2018-09-10 13:51:28.000	No	1536	1024
2	/Users/daniel/Downloads/ds1/2.fit	120.000000	23.883333	25.883333	2018-09-10 14:15:21.000	No	1536	1024
3	/Users/daniel/Downloads/ds1/3.fit	120.000000	28.250000	54.133333	2018-09-10 14:43:36.000	No	1536	1024
4	/Users/daniel/Downloads/ds1/4.fit	120.000000	28.266667	82.400000	2018-09-10 15:11:52.000	No	1536	1024

Figure 14 - Image Manager with Four Images Loaded

Step 2: View the Images

It is always good practice to verify the quality of the images. To do this, navigate to ***Action->View Images*** from the main menu. This will open the “Image Viewer” window. Now, if you go back to the “Image Manager” window, you can click on the different images and see them displayed in the “Image Viewer” accordingly. You will note that they are not yet aligned as there is some noticeable shift from one from image to the next. But the quality of the images is good, with stars having acceptable focus.

Step 3: Calibrate the Images

Now that you have visually inspected the images, it is time to prepare them for processing. The first step in this procedure is calibration. Navigate to ***Action->Calibrate Images*** from the main menu and you will see a new window appear for calibration settings.

Because these images have already been dark subtracted and flat fielded, it is not necessary to perform those steps here. However, when you are calibrating your own images, be sure to apply the appropriate dark frame and flat frame sources. “Pseudo flat” is also an option if you do not have a flat frame available. But for this dataset, the only calibration required is to normalize the images. This is important when using the synthetic tracker (as shown in later examples). So, choose the option “Normalize Images”. However, when doing sensitive photometry, it is better not to use normalization for best results.

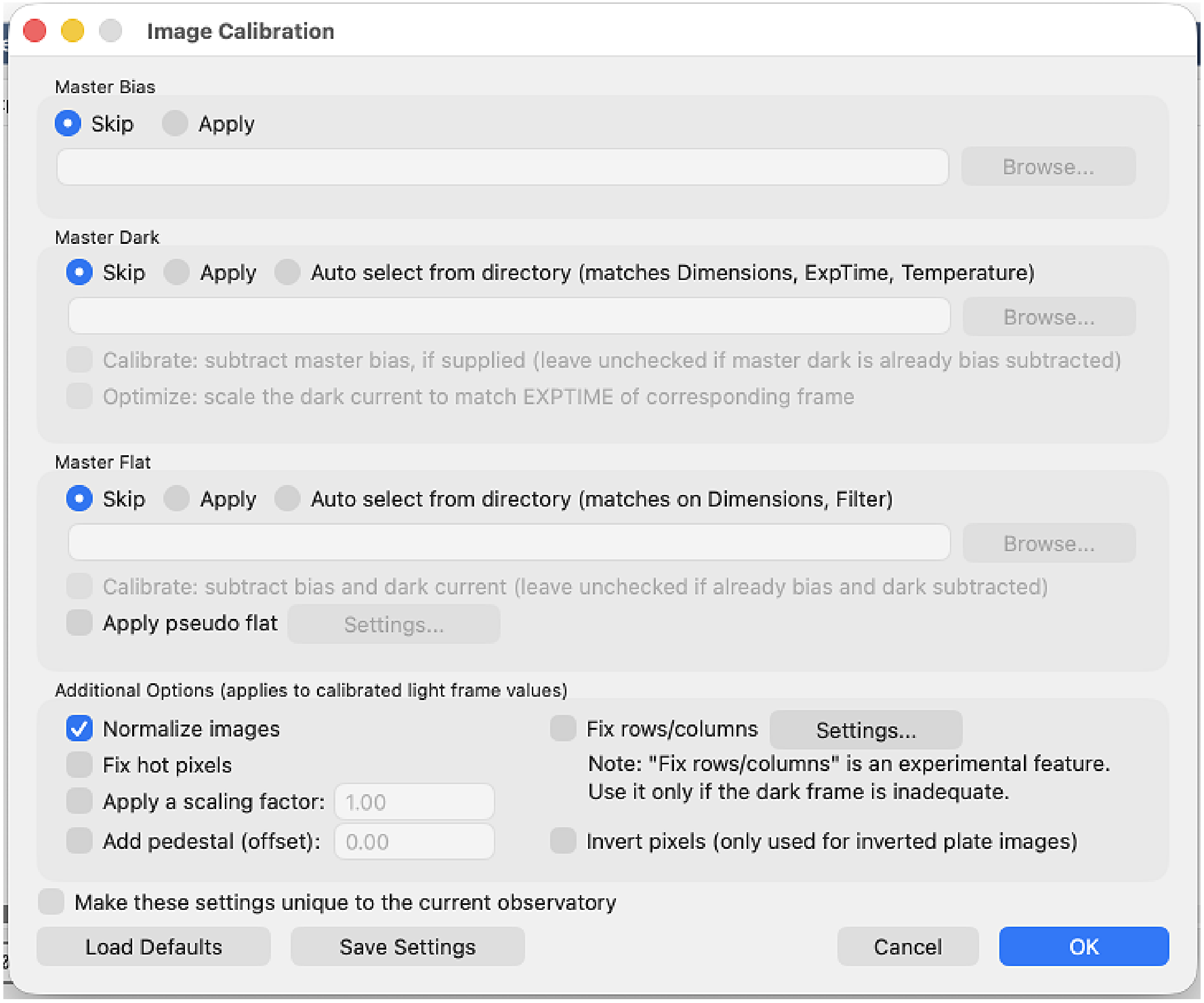


Figure 15 - Calibration Settings

Next, click "OK" to proceed with calibration. The resulting images will be saved in a new folder labeled "ds1_c". Navigate to that directory and load the now-calibrated images. Note: You can also have Tycho automatically load the output files. Navigate to **Settings->Image Directories**, and check the option labeled "Auto load output files into Image Manager".

Step 4: Plate Solve the Images

Navigate to the calibrated images, contained in directory "ds1_c". Load them as before. Now proceed to ***Action->Plate Solve Images***.

The images have a known pixel scale of around 1.65"/pixel. However, you can choose a wide lower- and upper-bound for the pixel scale, as shown in Figure 16. For more details, refer to the section labeled "Configure the Plate Solver". Proceed to plate solve the images by clicking the "Start" button.

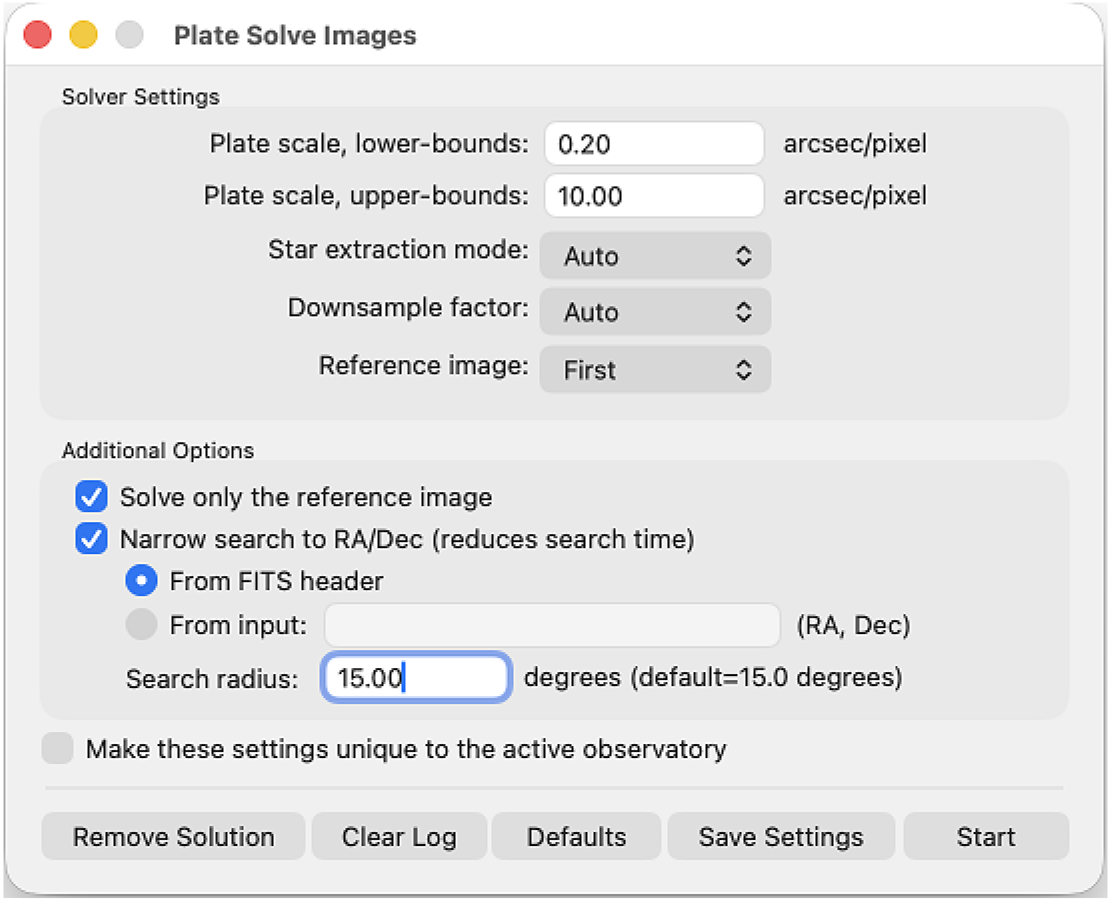


Figure 16 - Plate Solve Images

When the solver has finished, the images will remain loaded in the “Image Manager” – no new output directory is generated at this step. It is good practice to validate the plate solve by going to ***Action->View Images***, and checking that the catalog stars (blue boxes) are overlaid on top of the actual stars. To do this, choose ***File->Load Star Catalog*** from the menu of the “Image Viewer”.

If you do not see any stars, make sure ***Display->Catalog Stars*** is checked (from Image Viewer menu), and that you have configured the Star Catalog settings. If you have not configured the star catalog, now is a good time to do so. Refer to the section labeled “Configure the Star Catalog” for more details.

Step 5: Align the Images

The next step is to align the images. Proceed to ***Action->Align Images*** and choose the desired alignment options. Refer to Figure 17 for an example. Then click “OK” to proceed. As before, the images will be saved to a new directory, this time labeled “ds1_c_a”.

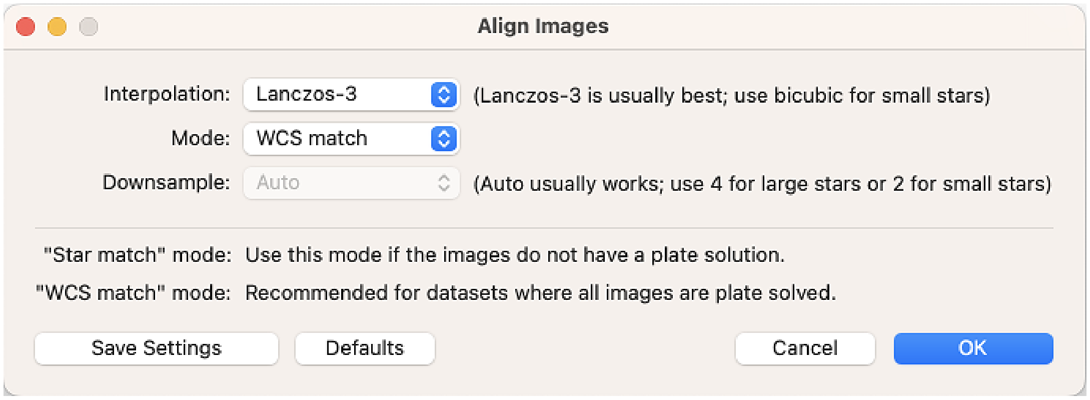


Figure 17 - Align Images

TIP: Using "Express Mode"

As you have observed, it can be rather tedious to perform each of the above steps manually. For this reason, there is a feature called "Express Mode", which streamlines the process. If you would like to try "Express Mode", load the original (uncalibrated, unaligned) images into the "Image Manager", and then navigate to ***Action->Express Mode*** from the main menu.

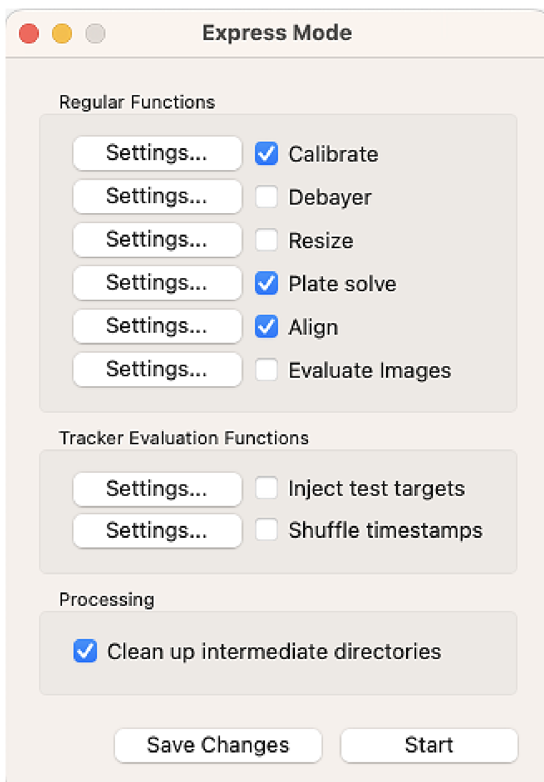


Figure 18 - Express Mode

As you can see, "Express Mode" offers the ability to carry out each step in an automated and streamlined fashion. Once you have the correct settings for each step, click the "Start" button and the program will automatically perform each step. When it is finished, the results are stored in a single output directory with the appropriate labeling.

Step 6: Verify Observatory Settings

Now that the images have been calibrated, plate solved, and aligned, it is time to start generating some measurements (also known as “observations”). Load the processed images, and then navigate to ***Action->View Images*** from the main menu.

Important: Before you proceed, you will need to change your observatory settings to match those of the observatory that acquired these images. In this example, the observatory that acquired these images has Minor Planet Center (MPC) code “Q62”, located in Siding Spring, Australia. To make this change, navigate to ***Settings->Observatory*** from the main menu and choose ***Action->Add Observatory*** from the window that appears. Give the observatory a label such as “Siding Spring” and then choose “Already have an MPC code” for the observatory status. Specify “Q62” for the MPC code, and click the button “Apply location from MPC code” to automatically populate the location details. For the telescope section, you can populate the details as shown in Figure 19. Then click the “Next” button and make sure to use the same settings as shown in Figure 20. Then, click “Finished”.

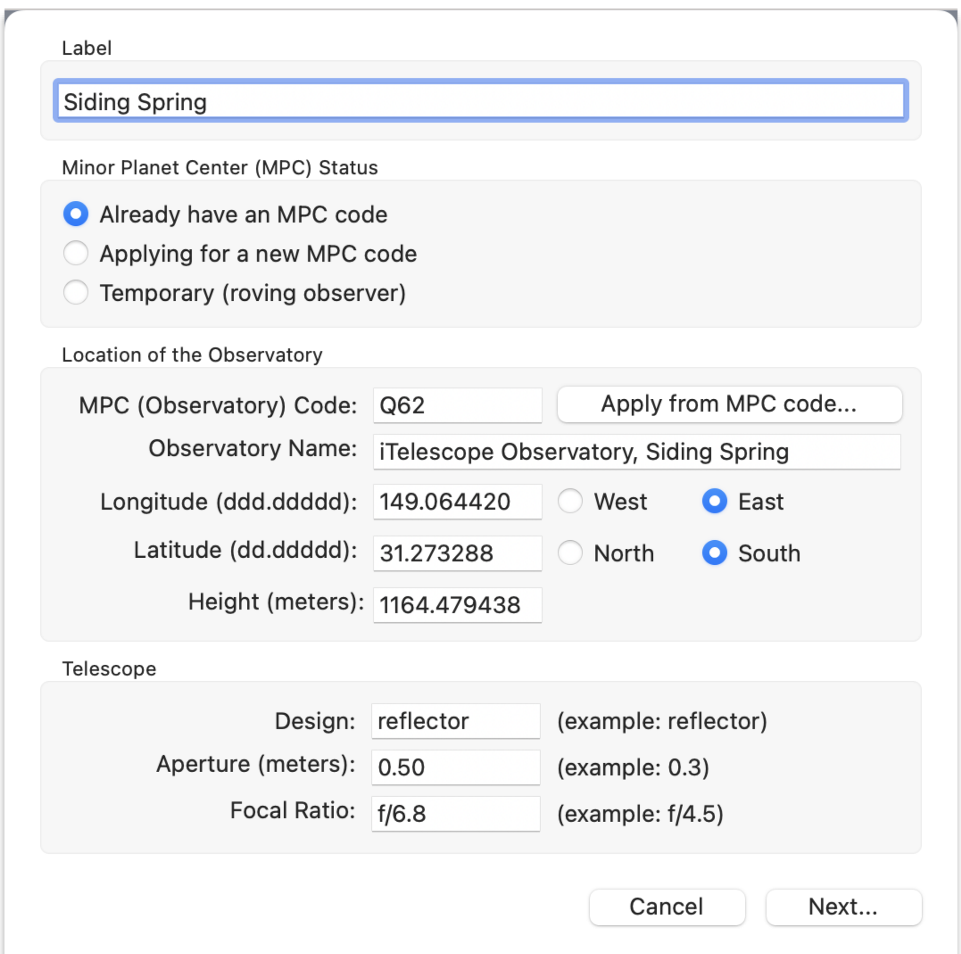


Figure 19 - Adding Observatory, Page 1/2

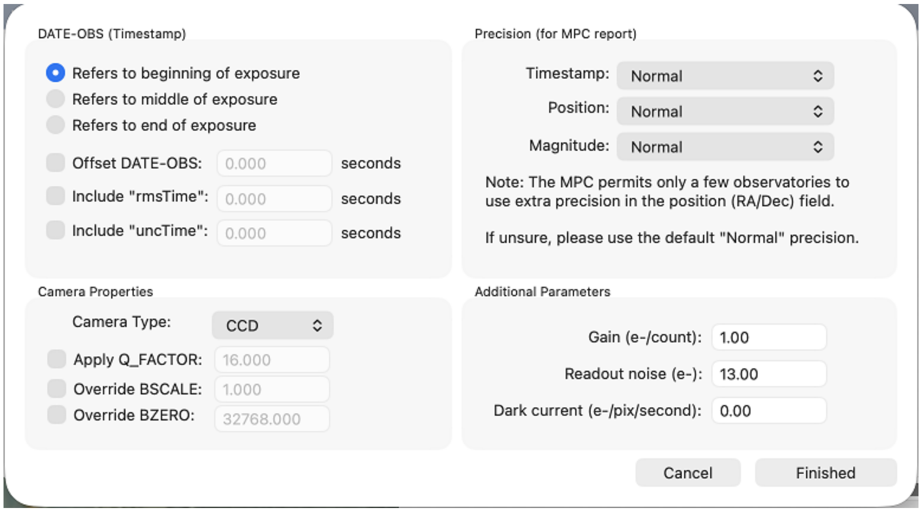


Figure 20 - Adding Observatory, Page 2/2

Now that “Siding Spring” has been added as an observatory, proceed to make it the “Active Observatory” by right-clicking on it in the list and choosing “Make Active”.

Step 7: Load Known Objects

Back at the “Image Viewer”, you can now proceed to have Tycho show known objects in the field by going to ***File->Load Known Objects*** from the “Image Viewer” menu. This feature depends on having configured the Known Objects database as well as the Find_Orb interface. Refer to the sections labeled “Configure the Find_Orb Software” and “Configure the Known Object Database” for more details.

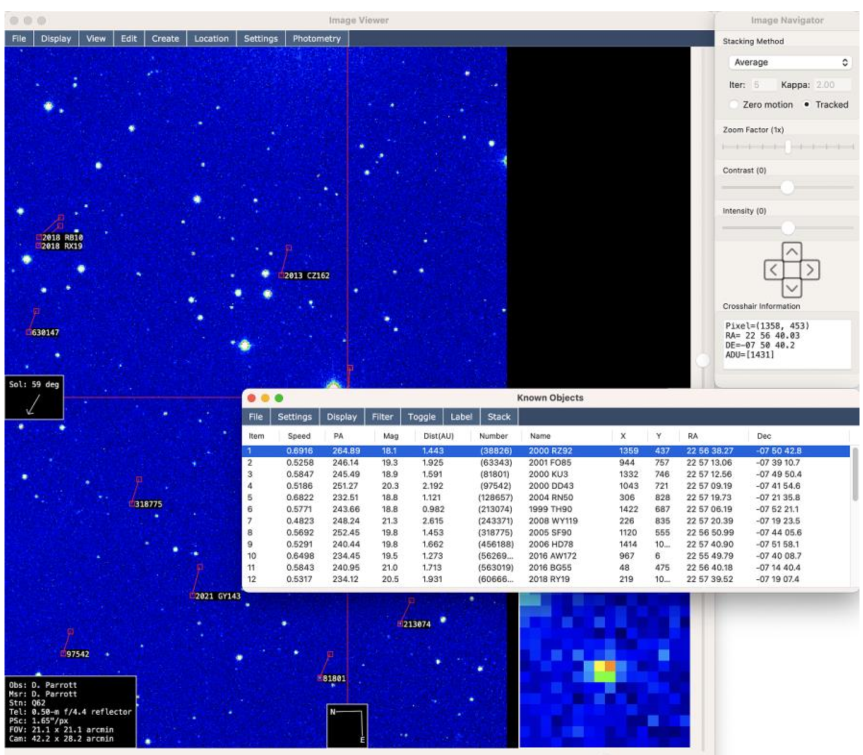


Figure 21 - Known Objects

At this point, a window should appear with a list of all known objects for this field. As you click on each item in the list, the “Image Viewer” will automatically update to the location of the object. If the object does not appear centered in the crosshairs, then you may need to verify either the observatory settings as described earlier (this dataset was acquired with MPC code Q62), or the “Known Objects” settings (*Settings->Known Objects* from the main menu). In rare situations, some cameras may specify a DATE-OBS that is not in UTC time; in these scenarios, you can specify the time offset to correct for this.

Step 8: Create and Verify Track

As an example, right-click on object with “Number” (permanent ID) of “(38826)”, also known as “2000 RZ92” (its provisional ID). When you have right-clicked the object, choose “Add to Track Navigator”. You can also do this with multiple objects at a time, but for now focus on just this object.

With the object added to the “Track Navigator”, you can either double-click it, or right-click and choose “Verify Track”. An animation of the object should now be shown in the “Image Viewer”, showing the movement of the asteroid.

The “Verify Track” window will limit to 3 observations by default. Generally, even if you had 60 images loaded, it is good practice to limit to 3 observations. The reason for this is that Tycho can then use 20 images for each observation, resulting in a much higher signal-to-noise ratio (SNR) for each observation. But if you want more observations, check the box “Advanced mode” and the drop-down list will allow you to have as many observations as there are images. In this example, proceed to generate three observations as shown in Figure 22.

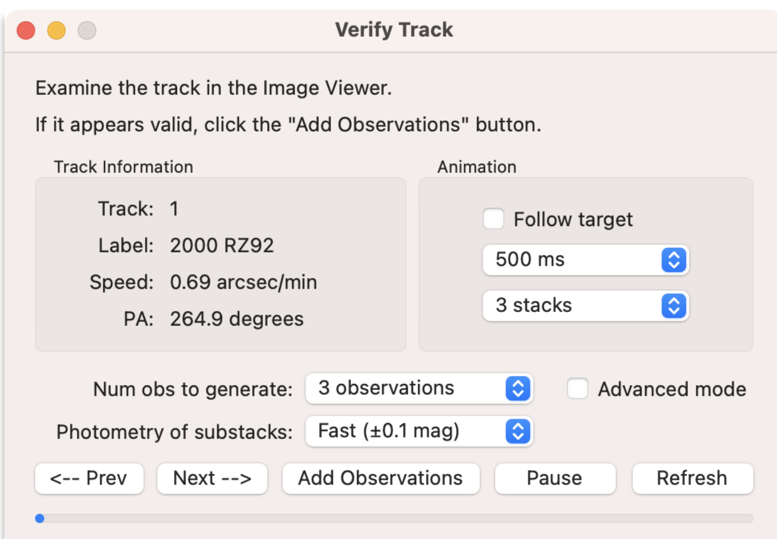


Figure 22 - Verify Track

Step 9: Create Measurements

Click the “Add Observations” button to continue. The “Object Designation” window will now appear, prompting for the designation of the object. Since this is a known object, the fields should already be populated.

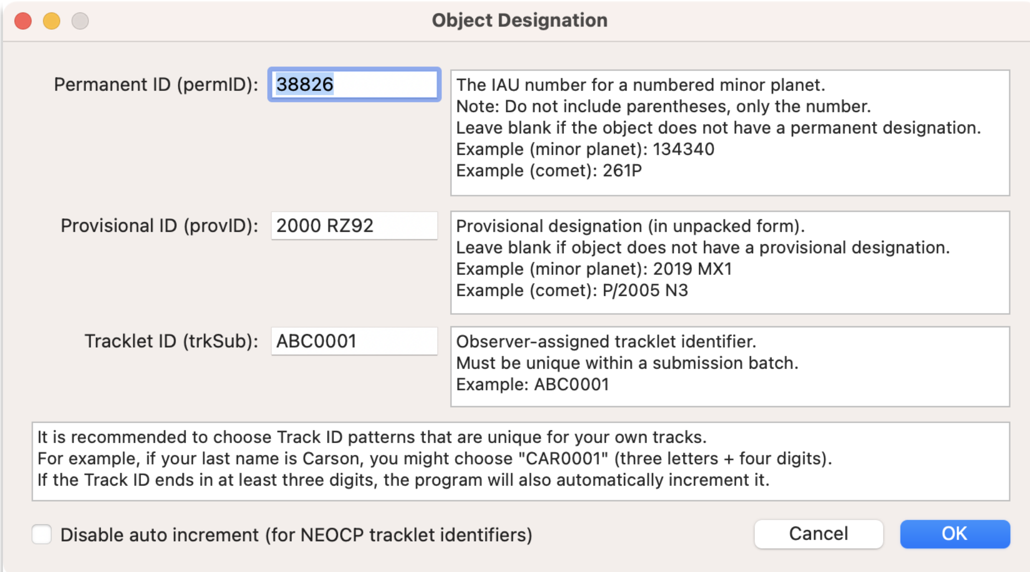


Figure 23 - Object Designation

Click "OK" to proceed. The measurements (observations) have now been generated for this object. At this point, you will want to verify that the observations were correctly generated by clicking on each measurement in the "Observations – All Targets" window.

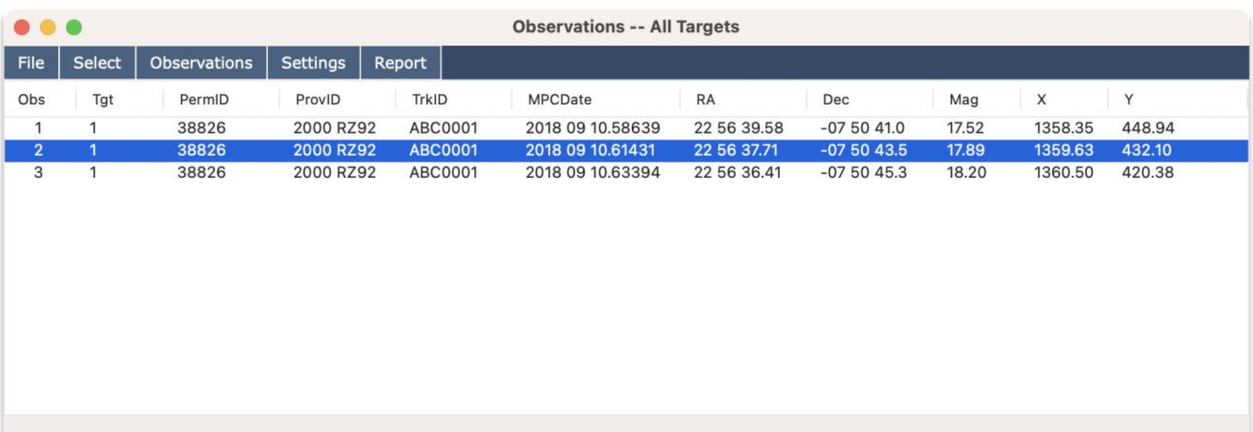


Figure 24 - Observations -- All Targets

As you click on each observation, the "Image Viewer" window is updated to show the centroid of the observation. You will note that the first observation uses a stack of the first two images.

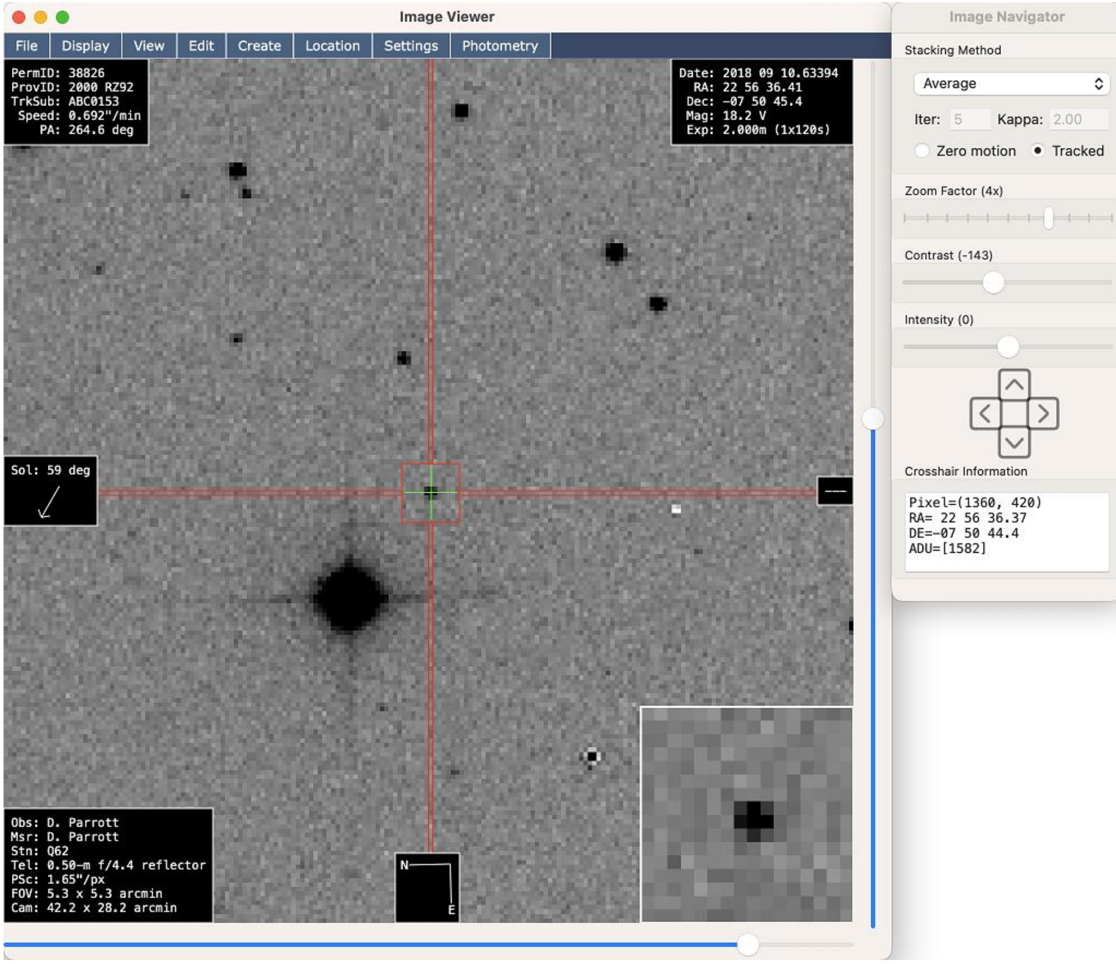


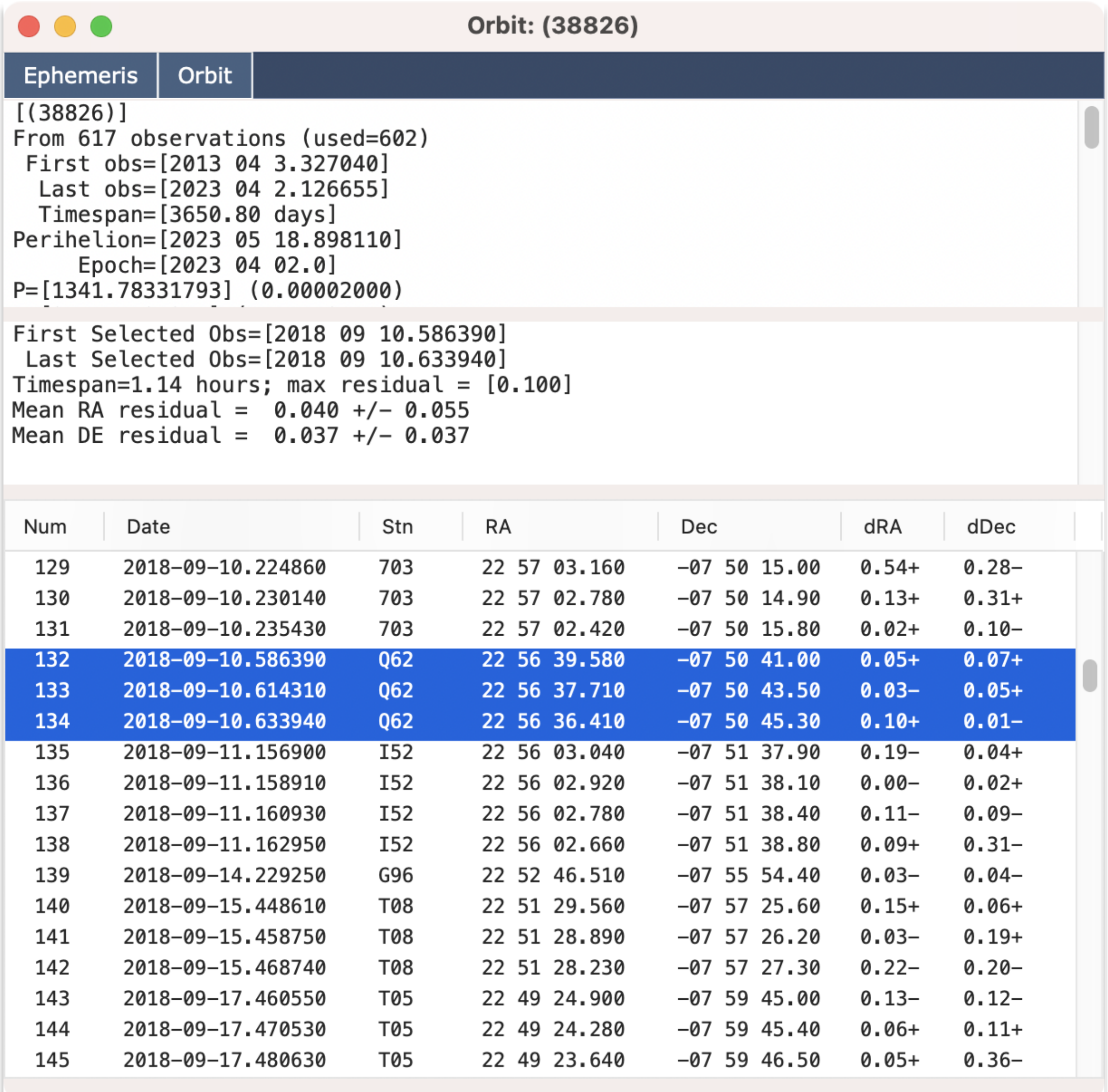
Figure 25 - Visual Inspection of Observations

Step 10: Validating the Measurements

You can validate the measurements by selecting them, then right-click and choose "View with Published Observations" from the popup menu that appears. It may take a moment, as the observations from other observatories are downloaded. When it is finished, a new window appears showing all of the observations for this object, with the ones that you just created at the very bottom.

As measurements of this object were already submitted by Q62, you will want to remove those observations from the list before doing the comparison. You will note the observations have a date of 2018 09 10, so scroll up in the list to around that timeframe and remove the observations that have Q62 in the far-right column. There should be about six observations to be removed.

When ready, click on "Compute Orbit" (the button in the lower-right corner of the window), and a new window will appear showing the residuals for these three measurements that you just generated.



Num	Date	Stn	RA	Dec	dRA	dDec
129	2018-09-10.224860	703	22 57 03.160	-07 50 15.00	0.54+	0.28-
130	2018-09-10.230140	703	22 57 02.780	-07 50 14.90	0.13+	0.31+
131	2018-09-10.235430	703	22 57 02.420	-07 50 15.80	0.02+	0.10-
132	2018-09-10.586390	Q62	22 56 39.580	-07 50 41.00	0.05+	0.07+
133	2018-09-10.614310	Q62	22 56 37.710	-07 50 43.50	0.03-	0.05+
134	2018-09-10.633940	Q62	22 56 36.410	-07 50 45.30	0.10+	0.01-
135	2018-09-11.156900	I52	22 56 03.040	-07 51 37.90	0.19-	0.04+
136	2018-09-11.158910	I52	22 56 02.920	-07 51 38.10	0.00-	0.02+
137	2018-09-11.160930	I52	22 56 02.780	-07 51 38.40	0.11-	0.09-
138	2018-09-11.162950	I52	22 56 02.660	-07 51 38.80	0.09+	0.31-
139	2018-09-14.229250	G96	22 52 46.510	-07 55 54.40	0.03-	0.04-
140	2018-09-15.448610	T08	22 51 29.560	-07 57 25.60	0.15+	0.06+
141	2018-09-15.458750	T08	22 51 28.890	-07 57 26.20	0.03-	0.19+
142	2018-09-15.468740	T08	22 51 28.230	-07 57 27.30	0.22-	0.20-
143	2018-09-17.460550	T05	22 49 24.900	-07 59 45.00	0.13-	0.12-
144	2018-09-17.470530	T05	22 49 24.280	-07 59 45.40	0.06+	0.11+
145	2018-09-17.480630	T05	22 49 23.640	-07 59 46.50	0.05+	0.36-

Figure 26 - Orbit Results

As you can see, the measurements are quite good, with "0.10" being the highest residual. Generally, any measurement with a residual below 1.50 is acceptable, but it is preferred to have residuals below 1.00. Several factors can influence the quality of measurements, including:

1) Accuracy of time source (especially important with fast-movers)
2) Star catalog
3) Quality of images

Step 11: Generating the Measurement Report

Now that you have validated the measurements, you can proceed to generate a report that could be used for submission. Note that these measurements should not be submitted as they are only for example purposes.

To generate the report, navigate back to the “Observations – All Targets” window. Then choose *Report->Generate MPC1992 Report* if you wish to generate an MPC1992 report. Otherwise, if you wish to generate an “ADES” report, choose *Report->Generate ADES Report*.

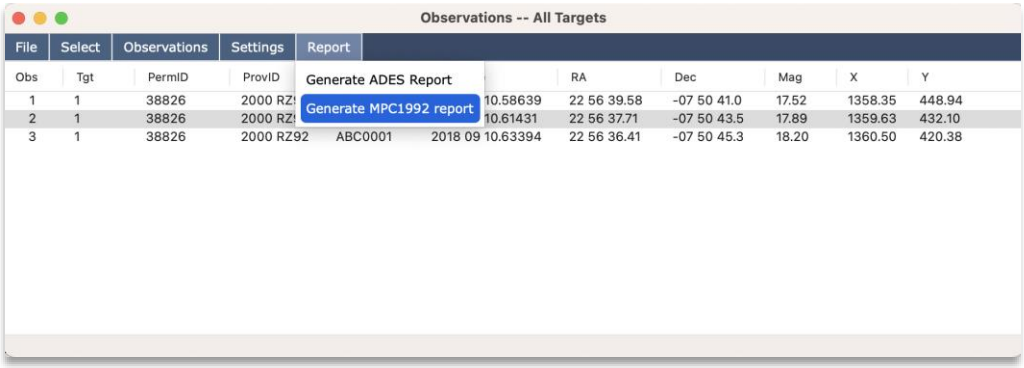


Figure 27 - Generating the Measurement Report

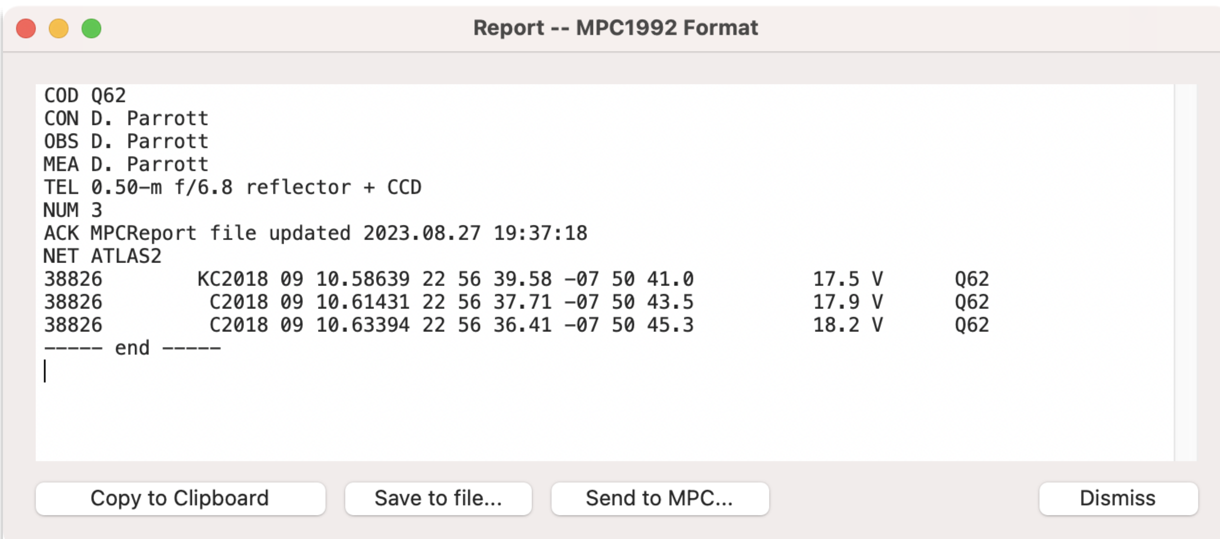


Figure 28 - MPC1992 Report

As you can see, the three observations are shown in an MPC1992 report. The first observation, being a stack of two images, has the “K” attribute next to the date.

Example #2: Measuring an “Unknown” Asteroid (using only 4 images)

This example expands upon the first example in that it assumes you now know how to calibrate, plate solve, and align. If not, please consult the first example for details on how to carry out those steps.

This example will show how to generate measurements of an “unknown” asteroid. In fact, the asteroid is already known, but for this example we will pretend otherwise. The purpose of this example is to show how to measure an asteroid without relying on the “Known Object” functionality.

Step 1: Load and Process Images

As before, proceed to calibrate, plate solve, and align the images. The images are the same as that in the first example, using dataset "ds1". Once you have finished processing the images, load them up and launch the "Image Viewer" by going to ***Action->View Images*** from the main menu.

Step 2: Blink Images

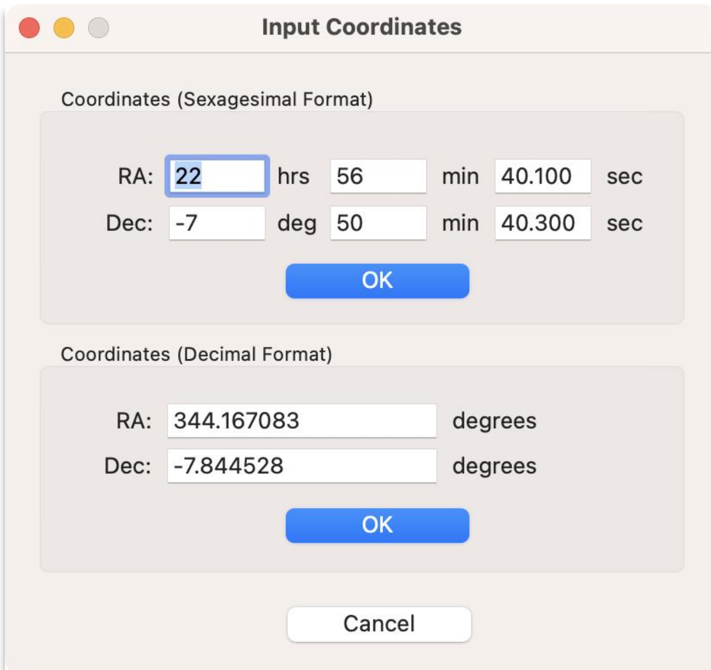


Figure 29 - Input RA/Dec Coordinates

From the "Image Viewer" menu, choose *Location->Center on RA/Dec*. A new window will appear, prompting for RA/Dec coordinates. You may either input the RA/Dec in the sexagesimal format, or in decimal format. For this object, its coordinates on the first image are as follows:

```
RA =  22 56 40.1
DE = -07 50 40.3 (note the "-07" is negative 07 degrees).
```

Click the "OK" button within the "Coordinates (Sexagesimal format)" section and the "Image Viewer" will be updated to display the image at those coordinates. You will notice that this looks similar to the asteroid measured in the previous example, and it is in fact the same one. But this time we will be measuring it in a different fashion.

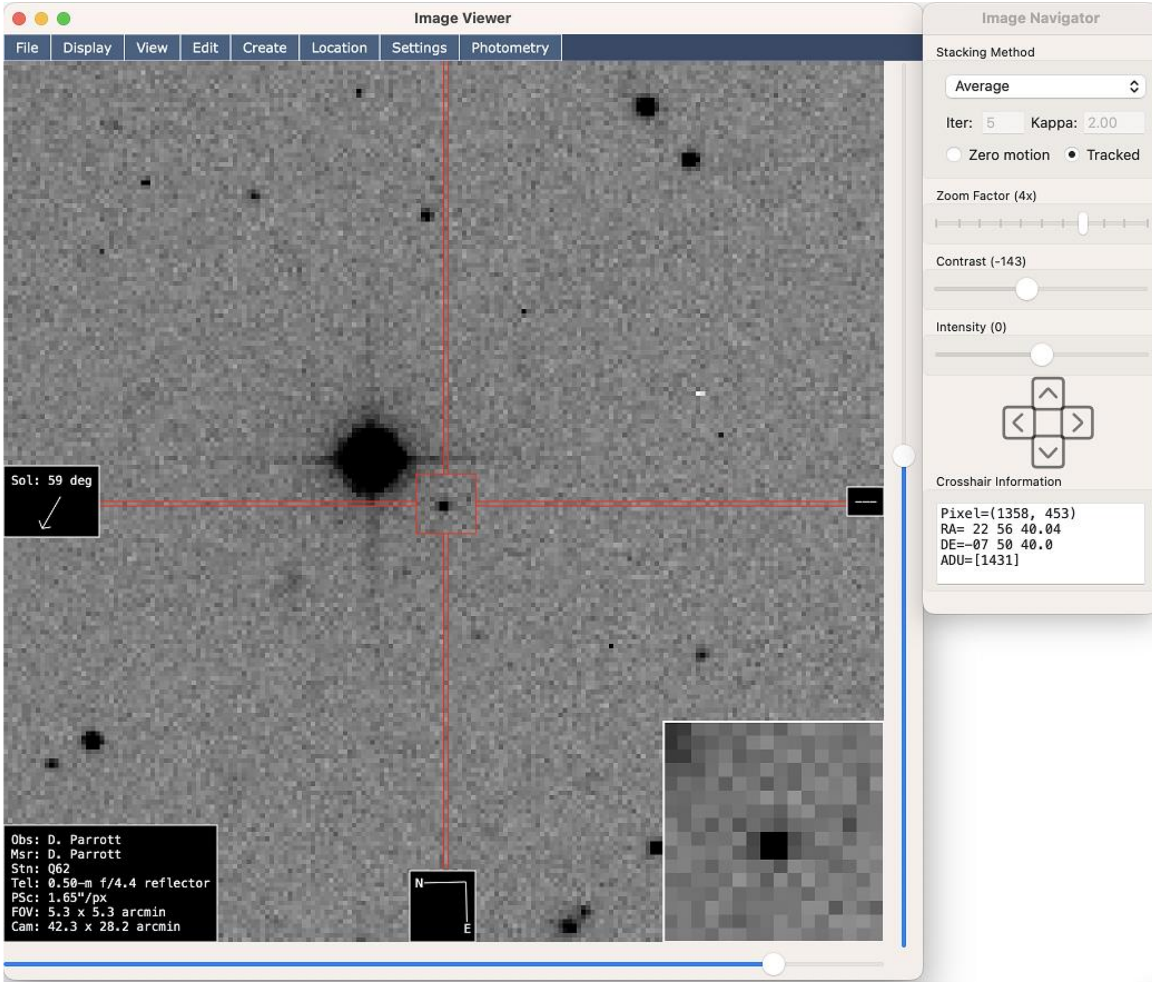


Figure 30 - Asteroid (38826)

Again, pretending that this is an "unknown" asteroid, blink through the images to reveal the moving object. You can either scroll through the images in the "Image Manager", or you can have the images blink in an automated fashion. If you wish to have them blink automatically, you can right-click inside the "Image Viewer" and choose "Create Track – From Current Position". In this particular scenario, it is not necessary to worry about the "current position", right-clicking anywhere inside the image should be fine. Since no motion has been defined, the new track will have a motion of zero, which allows the images to be blinked normally. Once the track has been added, double-click it to bring up the "Verify Track" window. Since you are just using this to blink images, you do not need to worry about creating observations yet. Also, in the "Animation" section, uncheck the box "Follow Target" and choose "4 stacks" so that each image is included separately in the blink animation. As you blink the images, you will see the motion of the object. To stop the blinking, either click the "Pause" button or click on an image in the "Image Manager". New in v8: You can now animate from the Image Manager by choosing "*Animate->250 ms*" (or another interval). To stop the animation, navigate to "*Animate->None*".

Step 3: Create Markers

Proceed to click on the first image in the "Image Manager", then double-click on the object in the "Image Viewer" so that it is properly centered in the crosshairs (you may need to zoom in). Now, right-click, and choose “Create Marker 1”. Then, click on the last image in the “Image Manager”, and double-click on the object once more. This time, right-click and choose “Create Marker 2”.

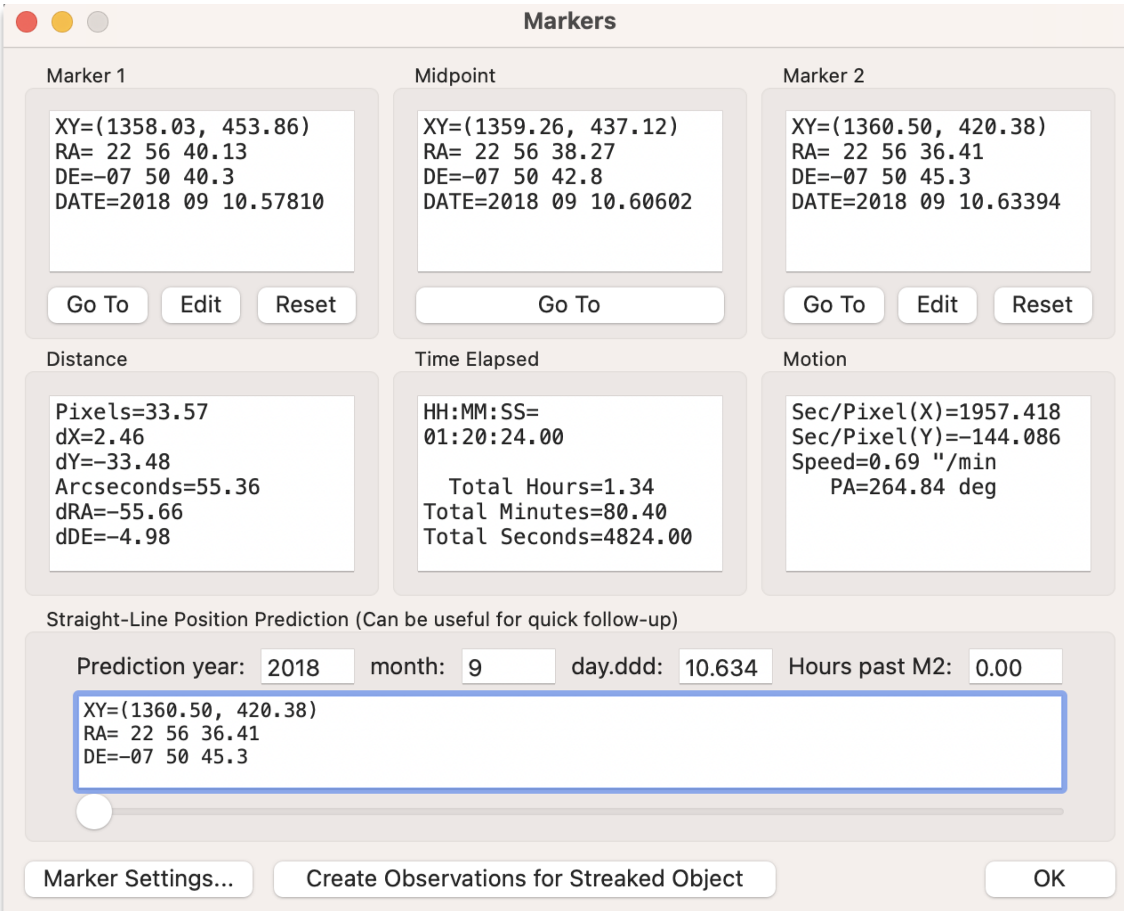


Figure 31 - Markers Window

Step 4: Create a Track

Having defined two markers to indicate the motion of the object, you can now create a track using these markers. This track will use the markers to compute the speed and position angle of the object. To create this track, right-click in the “Image Viewer” and choose “Create Track – From Markers”. You now have a second track, this time with speed and position angle (PA) of the object populated.

As before, you can double-click on the track and the “Verify Track” window will appear. But this time, the blinking animation will follow the actual object as it has now been programmed with its motion. So, if you check the box “Follow Target”, the object should remain centered within the crosshairs. At this point, you can proceed to create observations of the object, in a similar fashion to how you created observations in the first example.

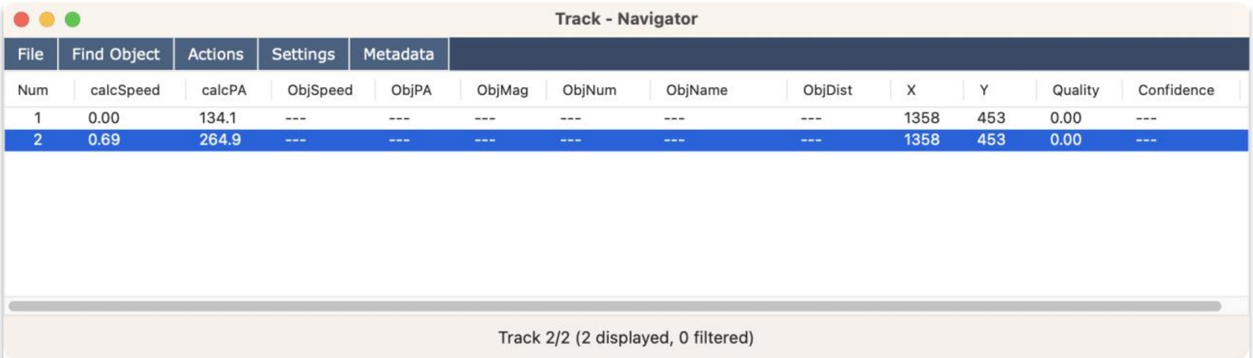


Figure 32 - Manually Created Tracks

Example #3: Creating Measurements Manually

Step 1: Center the Asteroid

With the images from the second example still open, go back to the "Image Manager" and click on the first image. Now, with the first image shown, go to the "Image Viewer" and double-click on the asteroid to center it (again, you may need to zoom in first before double-clicking on the asteroid).

Step 2: Create a Measurement

Now choose ***Create->Observation*** from the "Image Viewer" menu, or right-click inside the "Image Viewer" and choose "Create Observation" from the popup menu that appears. In either case, a new observation will be created and shown in a new window, "Observations – Single Target".

This approach allows you to manually create measurements without relying on the "Track Navigator". It also gives you more flexibility as you can choose what layer is used to generate the measurement: "median", "average", or "Abv" (detection layer). When you create a stack, these layers will all be different, with the "Abv" layer performing best at eliminating star interference and the "average" layer showing star trails. By default, the "Track Navigator" uses the "Abv" layer as it is usually best for automated generation of measurements and works well in crowded star fields; however, sometimes the "average" layer might be better, especially when the object has little movement.

Since you have not stacked any images, the layers will appear identical, as a single exposure is shown.

Step 3: Create a Stack (all images)

You have just now manually created a measurement using a single exposure. Now let's try creating a measurement using several exposures. Right-click in the "Image Viewer" and choose "Create Stack - Custom" from the popup menu that appears. This allows you to create a stack with a custom motion, either in terms of "Speed/PA" or in terms of "X/Y" motion. For this example, we will specify the motion of the asteroid with speed=0.7"/min and PA=265 degrees.

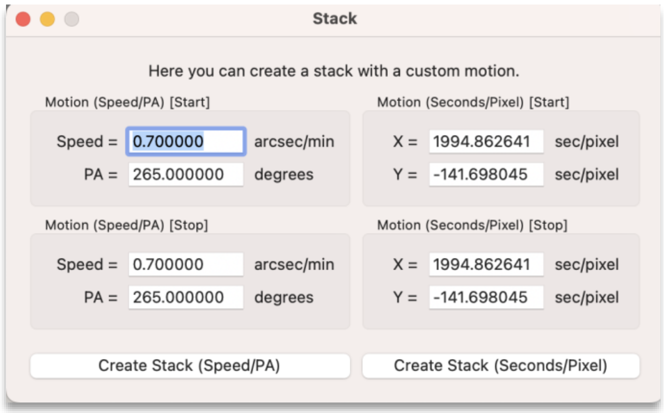


Figure 33 - Create Stack (Custom)

Now click the button “Create Stack (Speed/PA)”. You will note that the “Image Viewer” will be updated and will display a stack using median combine, average (mean) combine, or “Abv” combine. If you want to change the stack type (layer), then you can change the appropriate setting and click on “Create Stack (Speed/PA)” again. An example of the “Abv” layer is shown in Figure 34.

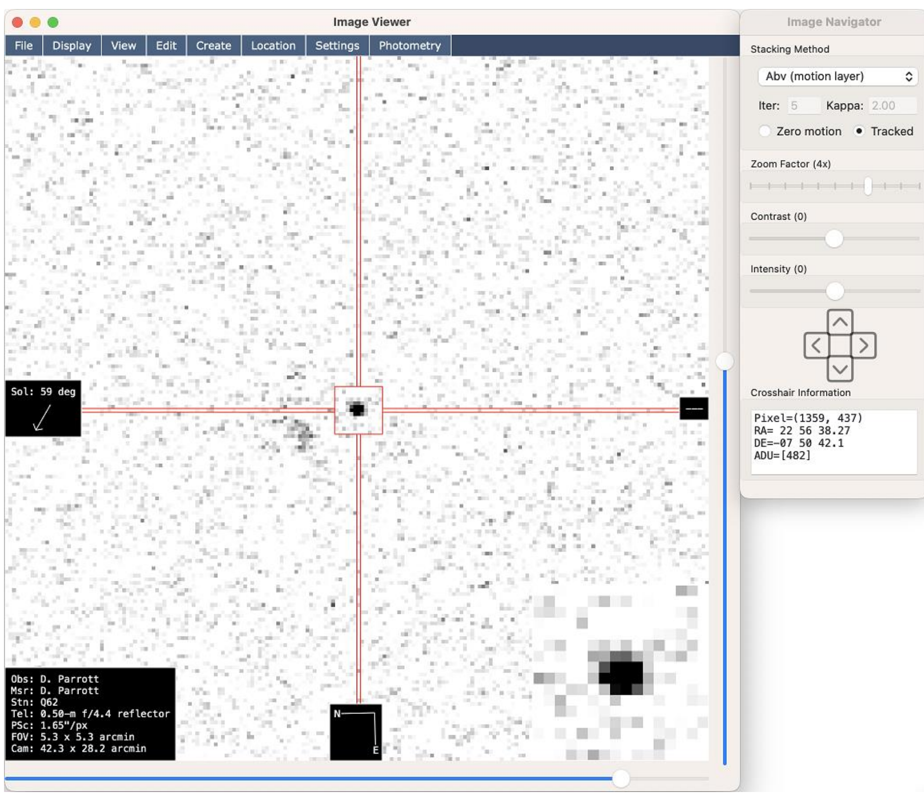


Figure 34 - "Abv" Stack

If the object is not centered in the crosshairs, double-click on it to center it. Then, right-click and choose “Create Observation”. You will now have a second observation listed, this time using all four exposures (ImgStart=1, ImgStop=4). Again, this is just an example; in practice, you would never create a measurement that reuses exposures from another measurement – ALWAYS use distinct and separate exposures for each measurement. The measurements may be generated from multiple exposures, but the exposures used by each measurement should be different from those used in other measurements.

Step 4: Create a Stack (subset of images)

As the previous step showed, if you create a stack with no images selected, it will create a stack using all of the images. This can be useful in certain scenarios when you want to stack all of the images to get the maximum SNR. But in this case, it also means that when you created the observation, you wound up with a measurement that uses all of the exposures – which you might not want. So, you may be wondering how to create a stack using only a subset of the images.

To do this, go back to the “Image Manager”, and select the last three images (the first image should not be highlighted, while the last three should be highlighted).

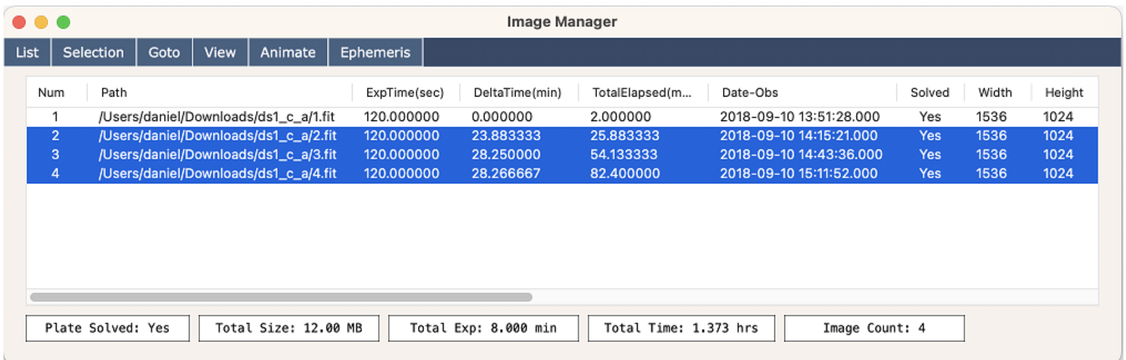


Figure 35 - Selecting the Last 3 Images

Then, go back to the “Create Stack – Custom” window (simply labeled “Stack”), and click on the button “Create Stack (Speed/PA)”. This time, when it creates the stack, it will do so using only the last three images. As you can see in Figure 36, when the stack is created with “average” layer, it is evident that three images were stacked because each star leaves behind three “footprints” in the image.

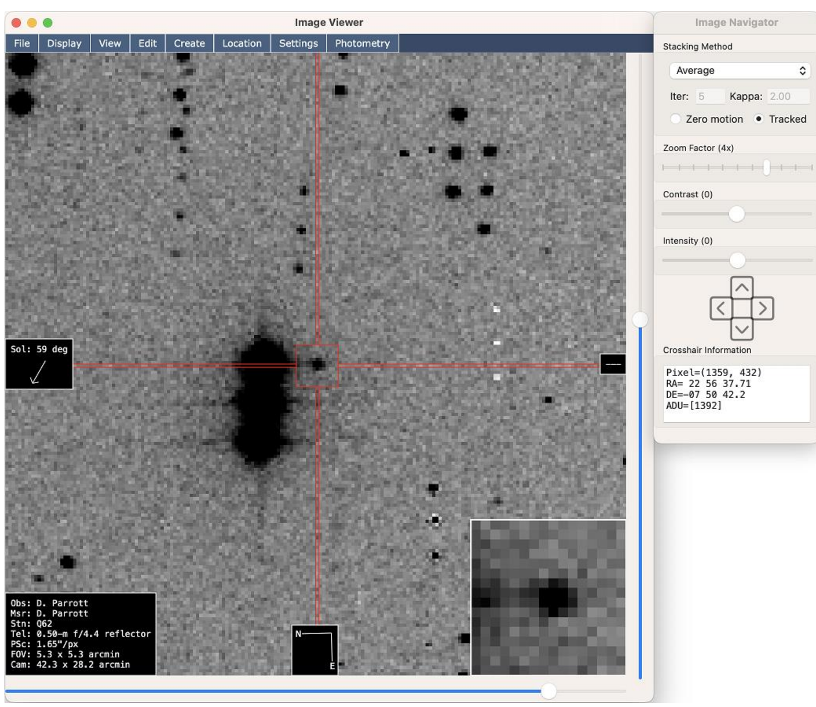


Figure 36 - Stack with 3 Images

Now that you have generated this stack of three images, you can create another observation. Double-click on the asteroid to center it, and then right-click and choose “Create Observation”. This time, you will note that “ImgStart=2” and “ImgStop=4”, meaning that images 2, 3, and 4 were used to generate this measurement. Proceed to delete the earlier measurement that used all of the images, which should leave you with two measurements total.

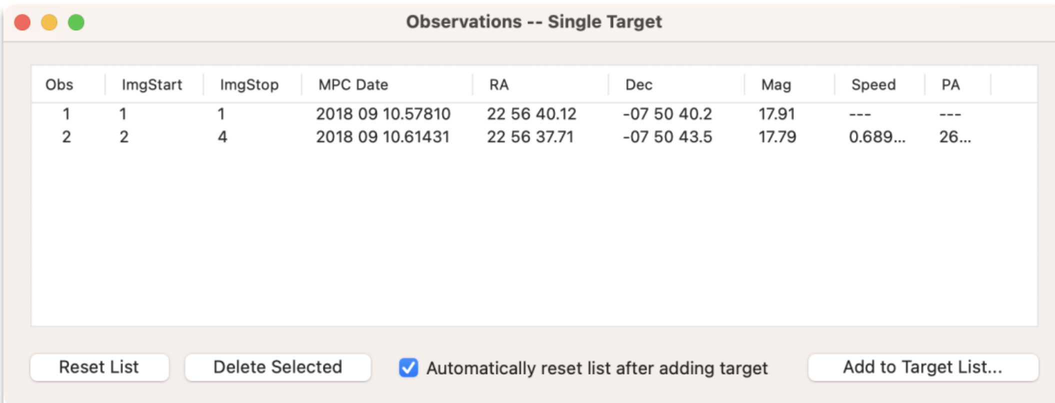


Figure 37 - Observations -- Single Target

Tip: In a later example, it will also be shown how to create these sub-stacks using the “Track – Positions” window, which is more convenient than manually creating each stack.

Step 5: Add to Target List

At this point, you have created measurements of some object. It could be an asteroid, a comet, a satellite, even just a star. The next step is to add these measurements of that object to the “Target List”, which allows them to be treated alongside measurements of other objects. So, proceed to click the “Add to Target List…” button. You will get a new window asking for the designation of the object. The permanent ID (permID) is 38826. Since it is a numbered object, there is no need to specify the provisional ID. Click “OK” to continue.

You will note the familiar window shown in the first example, “Observations – All Targets”. Here again, you could validate the measurements and generate a report. You could also click on each observation to see the green centroid indicating the exact center of the object. Generally speaking, it is not worthwhile to publish just two measurements of an object (3 measurements are preferred), but the purpose of this example was to show how to generate measurements in a manual fashion.

Step 6: Manually Adjusting the Centroid

Before going on to the next example, let’s try one more task. Here, we will pretend that one of the measurements had a poor centroid and requires manual correction. This is usually rare to occur, but if it happens, it is worthwhile to know how to fix it manually.

On the “Observations – All Targets” window, click on the second observation of the object. You will see the object centered with both the red crosshairs and the green centroid indicator. Again, let’s pretend that the green centroid needs adjustment, so right-click on the observation and choose “Modify Observation”. This will bring up a new window as shown in Figure 39.

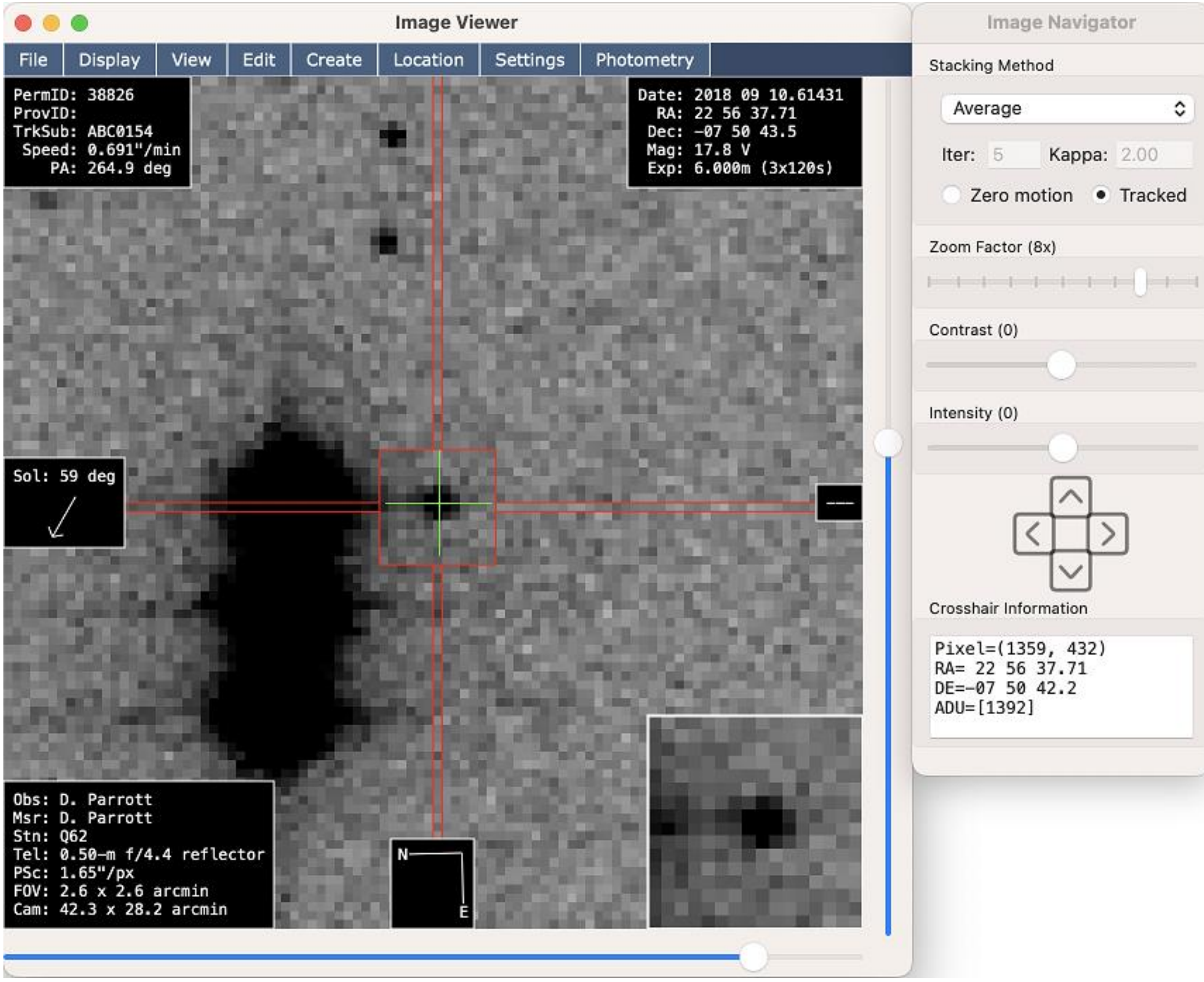


Figure 38 - Green Centroid Indicator

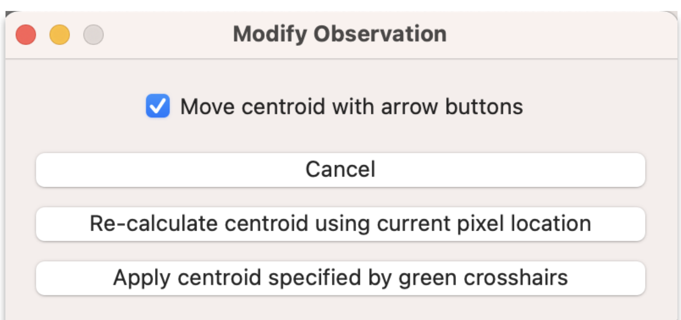


Figure 39 - Modify Observation

From the "Modify Observation" window, there are two ways to adjust the centroid. One is to move the centroid with fine adjustment (1/10th of a pixel). To do this, check the box "Move centroid with arrow buttons". Then click on the arrow buttons in the "Image Viewer" to move the centroid. When finished, click "Apply centroid specified by green crosshairs". You should see the "RA" and "DEC" of the observation be updated in the "Observations – All Targets" window when the button is clicked.
The other method to adjust the centroid is by specifying an initial pixel location from which Tycho will then compute a more accurate centroid. In other words, if the observation is "way off", the pixel

location can be used to get it back in the "ballpark" and then Tycho can re-compute the correct centroid. To use this method, uncheck the box "Move centroid with arrow buttons", and proceed to pan the image to where the object is located (even just double-clicking on the object). Then click the button "Re-calculate centroid using current pixel location". As with the other method, the "RA" and "DEC" of the observation should be updated in the "Observations – All Targets" window.

You can also use the "Modify Observation" window to change which layer is used for the observation. So, if you wanted to use the "Median" layer rather than "average" layer, you could open the "Modify Observation" window, double-click the object, and click "Re-calculate centroid using current pixel location". The centroid (as well as magnitude information) will then be recomputed using the specified layer.

Note: If you click on a different observation and the green centroid indicator does not follow it, that is likely because the "Modify Observation" window was kept open. Close the "Modify Observation" window when finished with it. It will also close by itself whenever you actually modify the observation, or whenever you click "Cancel".

Example #4: Measuring a Faint NEO (without synthetic tracker)

This example expands upon the first example in that it assumes you now know how to calibrate, plate solve, and align images. If not, please consult the first example for details on how to carry out those steps.

This example will show how to generate measurements of a faint NEO that is not detectable on individual frames. As such, unlike the previous examples that had a bright asteroid, this example makes use of many exposures in order to produce a sufficient signal-to-noise ratio (SNR).

Step 1: Load and Process Images

Load the dataset for this example, "ds2", and proceed to perform calibration, plate solving, and alignment. There should be 39 images in this dataset.

Step 2: Download Observations for the NEO

Navigate to ***Tools->Download Observations*** from the main menu. At the prompt, type in "2018 RB" for the object name, and choose the option "From obs database (confirmed object)". Then click "OK" to proceed. After a moment, a new window will appear, "Text Form – Observations". This window should be populated with the observations for the NEO.

Step 3: Compute NEO Orbit

Click the button in the lower-right corner, labeled "Compute Orbit". After a moment, a new window presenting the orbit of the object will appear.

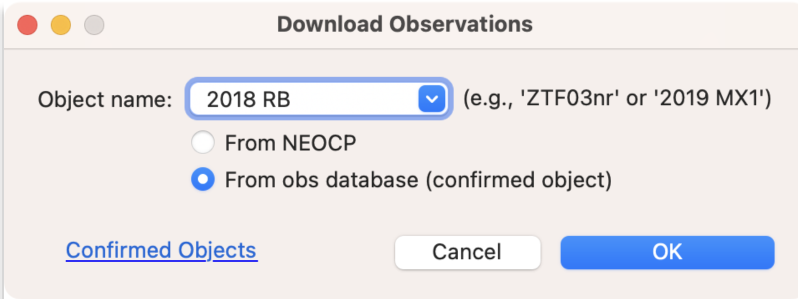


Figure 40 - Download Observations

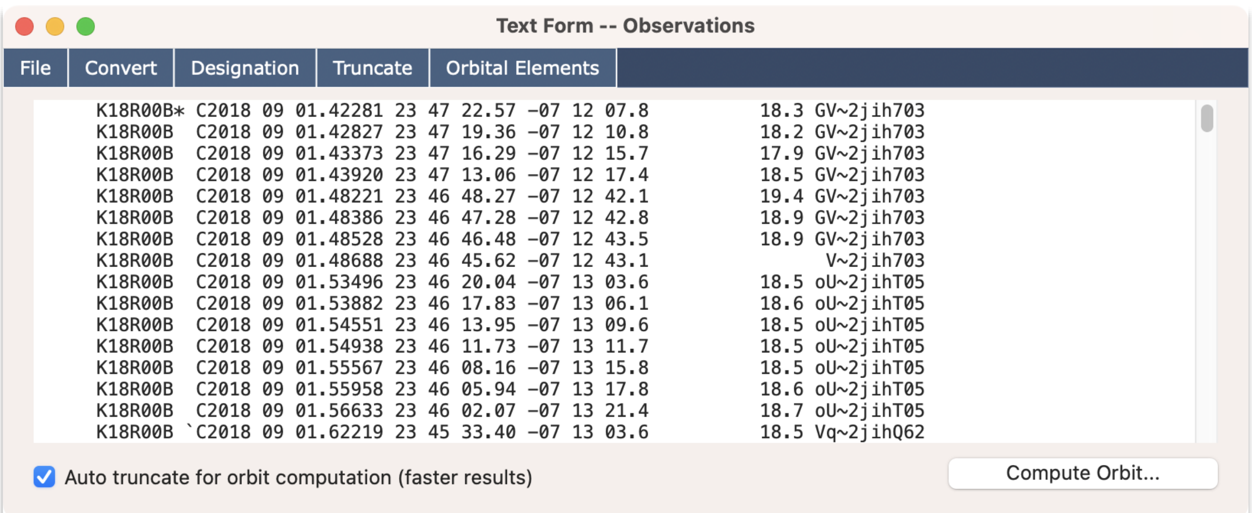


Figure 41 - Text Form -- Observations

Step 4: Attach Ephemeris to the Dataset

From the orbit window (titled “Orbit: 2018 RB”), navigate to ***Ephemeris->Attach to Dataset***. After a moment, you will note that the “EPH_” columns in the “Image Manager” are now populated with information about the object. If you scroll to the far-right, you can see the “EPH_IN_FOV” column which indicates whether or not the object is expected to be in the field of view for each image. This can be helpful as you typically want to exclude images where it is not in the field of view. For this dataset, the object should be in the field of view of each image, so the column should indicate “Yes” for every image.

Step 5: Create Stack (using ephemeris)

Now that the ephemeris information has been attached to the dataset, proceed to view the images by going to ***Action->View Images*** from the main menu. Then, right-click inside the “Image Viewer” and choose “Create Stack – Ephemeris” from the popup menu appears. Alternatively, you can also choose the option by going to ***Create->Stack – Ephemeris*** from the “Image Viewer” menu. At this point, you will note that the “Image Viewer” has been updated with the display of a stacked image according to the computed motion of the object.

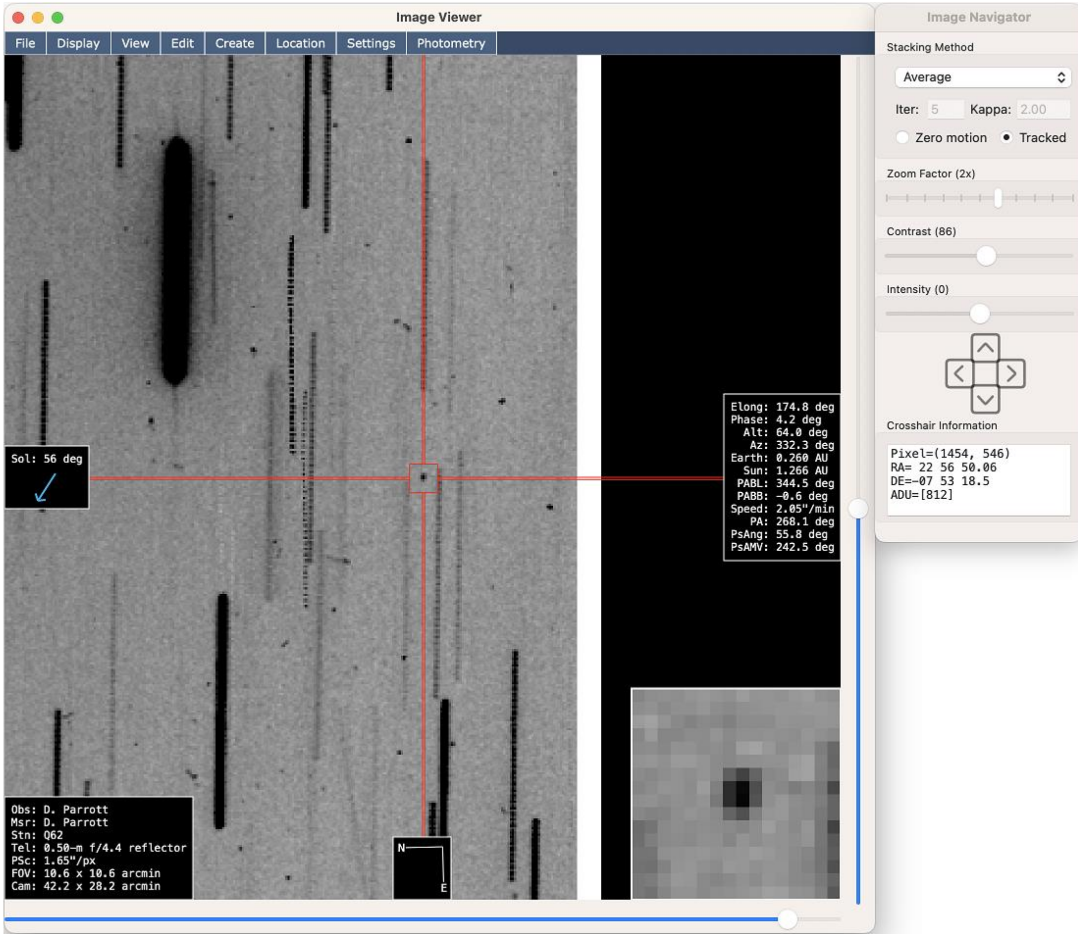


Figure 42 - Result of "Create Stack - Ephemeris"

In Figure 42, you can see the result of creating the stack. If your results look different, you may be using a different layer such as "Median" or "Abv". For this example, choose "average", and then recreate the stack to force an update of the display. Also, you can adjust the contrast to make the object stand out better. I chose (86) for the contrast slider. The median layer (as well as "Abv") does a better job at eliminating noise artifacts, while "average" layer can be useful for showing the star trails.

Step 6: Create a Track

Double-click the object to ensure that it is centered in the crosshairs (you may want to zoom in first). Then right-click and choose "Create Track – Current Position" from the popup menu that appears. This will add a new track to the "Track Navigator".

Step 7: Create Measurements

Double-click on the track and you can see the "Verify Track" window appear. Click the "Add Observations" button. Another window appears, "Object Designation", prompting for the designation of the object. Specify "2018 RB" (without the quotes) for the "Provisional ID (provID)" field. The object does not have a permanent designation, so leave (permID) blank. Click "OK" to continue.

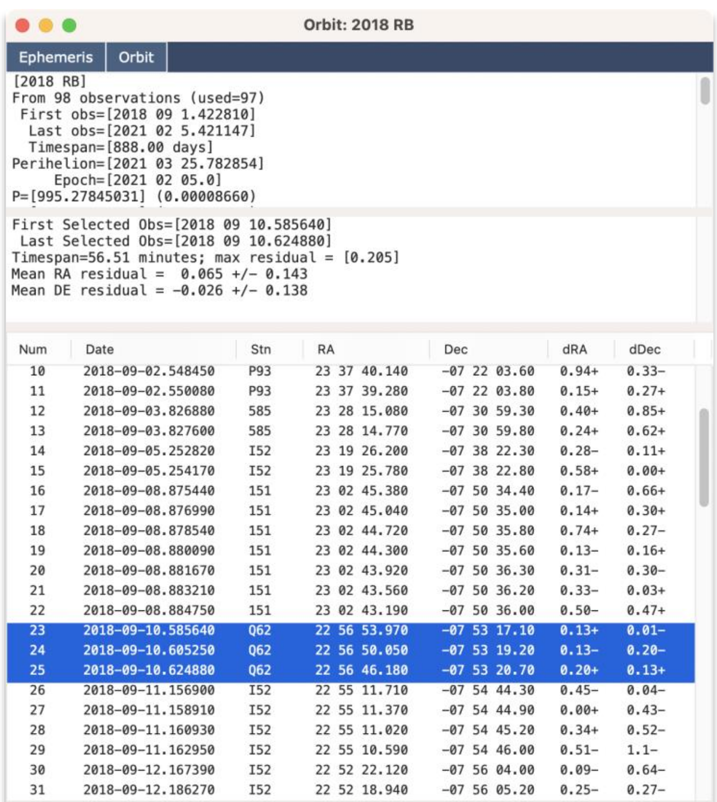
As before, you can perform a visual inspection of each observation by clicking on them in the "Observations – All Targets" window. On rare occasions you might also want to manually adjust the centroid; if so, please refer to the section labeled "Step 6: Manually Adjusting the Centroid" in the previous example. For this example, there should be no need to modify the centroid.

Step 8: Validate the Measurements

As described in the first example, you can validate the measurements by right-clicking on them and choosing "View with Published Observations". This will bring up the "Text Form – Observations" window, with the newly-created measurements at the very bottom. Again, the observatory that acquired these images is "Siding Spring" in Australia, with MPC code "Q62". So, if you are seeing a different MPC code shown in the far-right of the observation text, you will need to make "Siding Spring" the active observatory. Refer to the first example for details on how to do this.

Because Q62 already submitted observations of this object, you will want to remove those observations from the list before comparing your newly-created observations with those from other observatories. Scroll up the list until you reach 2018 09 10 and locate the observations by Q62. There should be around six observations; delete them and leave the others.

Now click the button "Compute Orbit" and you should see the residuals (errors) for the observations you just created. From this result, it appears that the highest residual is "0.20", which is quite good. Again, any measurement with a residual under 1.50 is "acceptable", but it is preferred to be under 1.00 when possible.


Orbit: 2018 RB

Ephemeris | Orbit

```
[2018 RB]
From 98 observations (used=97)
 First obs=[2018 09 1.422810]
  Last obs=[2021 02 5.421147]
  Timespan=[888.00 days]
Perihelion=[2021 03 25.782854]
     Epoch=[2021 02 05.0]
P=[995.27845031] (0.00008660)
```

```
First Selected Obs=[2018 09 10.585640]
 Last Selected Obs=[2018 09 10.624880]
Timespan=56.51 minutes; max residual = [0.205]
Mean RA residual =  0.065 +/- 0.143
Mean DE residual = -0.026 +/- 0.138
```

Num	Date	Stn	RA	Dec	dRA	dDec
10	2018-09-02.548450	P93	23 37 40.140	-07 22 03.60	0.94+	0.33-
11	2018-09-02.550080	P93	23 37 39.280	-07 22 03.80	0.15+	0.27+
12	2018-09-03.826880	585	23 28 15.080	-07 30 59.30	0.40+	0.85+
13	2018-09-03.827600	585	23 28 14.770	-07 30 59.80	0.24+	0.62+
14	2018-09-05.252820	I52	23 19 26.200	-07 38 22.30	0.28-	0.11+
15	2018-09-05.254170	I52	23 19 25.780	-07 38 22.80	0.58+	0.00+
16	2018-09-08.875440	151	23 02 45.380	-07 50 34.40	0.17-	0.66+
17	2018-09-08.876990	151	23 02 45.040	-07 50 35.00	0.14+	0.30+
18	2018-09-08.878540	151	23 02 44.720	-07 50 35.80	0.74+	0.27-
19	2018-09-08.880090	151	23 02 44.300	-07 50 35.60	0.13-	0.16+
20	2018-09-08.881670	151	23 02 43.920	-07 50 36.30	0.31-	0.30-
21	2018-09-08.883210	151	23 02 43.560	-07 50 36.20	0.33-	0.03+
22	2018-09-08.884750	151	23 02 43.190	-07 50 36.00	0.50-	0.47+
23	2018-09-10.585640	Q62	22 56 53.970	-07 53 17.10	0.13+	0.01-
24	2018-09-10.605250	Q62	22 56 50.050	-07 53 19.20	0.13-	0.20-
25	2018-09-10.624880	Q62	22 56 46.180	-07 53 20.70	0.20+	0.13+
26	2018-09-11.156900	I52	22 55 11.710	-07 54 44.30	0.45-	0.04-
27	2018-09-11.158910	I52	22 55 11.370	-07 54 44.90	0.00+	0.43-
28	2018-09-11.160930	I52	22 55 11.020	-07 54 45.20	0.34+	0.52-
29	2018-09-11.162950	I52	22 55 10.590	-07 54 46.00	0.51-	1.1-
30	2018-09-12.167390	I52	22 52 22.120	-07 56 04.00	0.09-	0.64-
31	2018-09-12.186270	I52	22 52 18.940	-07 56 05.20	0.25-	0.27-

Figure 43 - Validating the Measurements

Step 9: Generate Report

As described in the first example, you can now proceed to create either an MPC1992 report or an "ADES" report by going to ***Report->Generate MPC1992 Report*** or ***Report->Generate ADES Report*** from the "Observations – All Targets" window.

As before, do not submit these observations, as they are only for example purposes.

Step 10: Using "Track – Positions" to Generate Sub-stacks

While the images are still loaded, try experimenting with the "Track – Positions" window. This window should still be open from having created the track earlier. If not, you can go to *Window – Track – Positions* from the main menu to bring it to the front. If you do not see the window in the list of available windows, then you should try clicking on the track in the "Track Navigator". Or, worst case scenario, if the "Track Navigator" window is no longer open, proceed to re-create the track as described in the previous step "Step 6: Create a Track".

Now that you have the "Track – Positions" window in view, click on the first entry in the list. This will update the "Image Viewer" to center on the object in the first image. You may want to reset contrast back to "(0)" when examining individual images. With the first entry still highlighted in the "Track – Positions" window, use the "Down" arrow key on the keyboard to rapidly move to the other entries. You should now see the object appear to move in the "Image Viewer" as it follows the object on each image. From this, you can see that the object is indeed quite faint on each individual exposure.

To create a sub-stack, click the first entry in the "Track – Positions" window, then hold down the "SHIFT" key and click on the 13th entry. This selects entries 1-13. The "Image Viewer" will now display a sub-stack generated from images 1-13. At this point, you can now double-click on the object in the "Image Viewer" and create an observation by right-clicking and choosing "Create Observation" from the popup menu that appears. Then, to create another observation using images 14-27, go back to the "Track – Positions" window and click on entry 14, then hold down the shift key, and click on entry 27. Another sub-stack will be created, this time using images 14-27. As before, you can double-click on the object in the "Image Viewer" and create an observation by right-clicking, and choosing "Create Observation" from the popup menu that appears. Repeat once more with images 28-39 to create a third observation, if desired. As mentioned in a previous example, these observations are added to the "Observations – Single Target" window. You need click on the "Add to Target List…" button to move them over to the "Observations – All Targets" window.

Tip: If you are wondering what the "Add Animation Stack" button does on the "Track – Positions" window, it simply adds another stack to the blink animation. The blink animation is controlled by the "Blink Stacks" window, which is normally kept hidden.

Another feature of the "Track – Positions" window is that it enables you to create a sub-stack in real-time by clicking on an entry in the list and holding down the "SHIFT" key while pressing the "Down" arrow key on the keyboard. You should see the "Image Viewer" update in real-time as the images are combined. This is usually quite fast if you have "GPU Acceleration" enabled.

Example #5: Measuring a Faint NEO (with synthetic tracker)

The purpose of this example is to show how you can detect and measure a faint NEO using the synthetic tracker. Since the object is quite faint, many exposures are used. And as the synthetic tracker is used, a minimum of 11 images are required.

Step 1: Load and Process the Images

This example uses the same dataset as the previous example, "ds2". Proceed to load and process the images (calibrate, plate solve, and align). Refer to the first example for details on how to perform these steps.

Step 2: Attach Ephemeris to the Dataset

Similar to the previous example, you will retrieve observations of the object and use those observations to generate ephemeris information. Then, attach that ephemeris to the dataset. The object is again "2018 RB". Refer to the previous example for details on how to attach ephemeris information.

Step 3: Run the Synthetic Tracker

Navigate to ***Action->Synthetic Tracker*** from the main menu. A new window will appear, prompting for the "Sensitivity Threshold". A setting of 50% works well here, but if the object is extremely faint you may want to increase the sensitivity. A lower sensitivity can also be specified, resulting in faster searches at the tradeoff of potentially losing some detections.

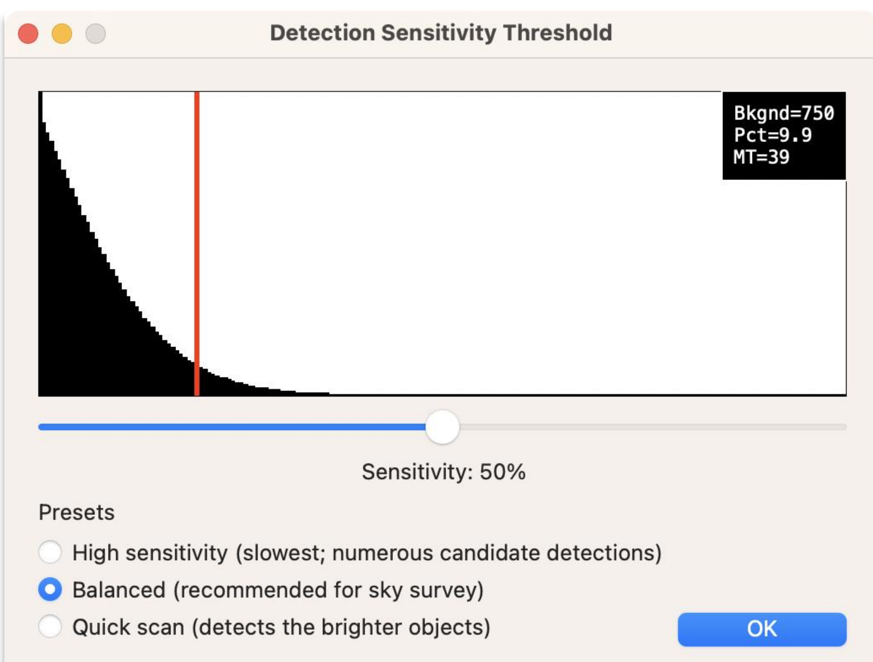


Figure 44 - Sensitivity Threshold

Proceed with the default of 50% sensitivity and click "OK" to continue.

On the next window, you are prompted for the search parameters. Since this is a search of a known object, and since you have attached ephemeris information to the dataset, you can click the "Use Dataset Ephemeris" button to automatically confine the search to the motion of the object. You should see that the speed and PA have been limited as shown in Figure 45.

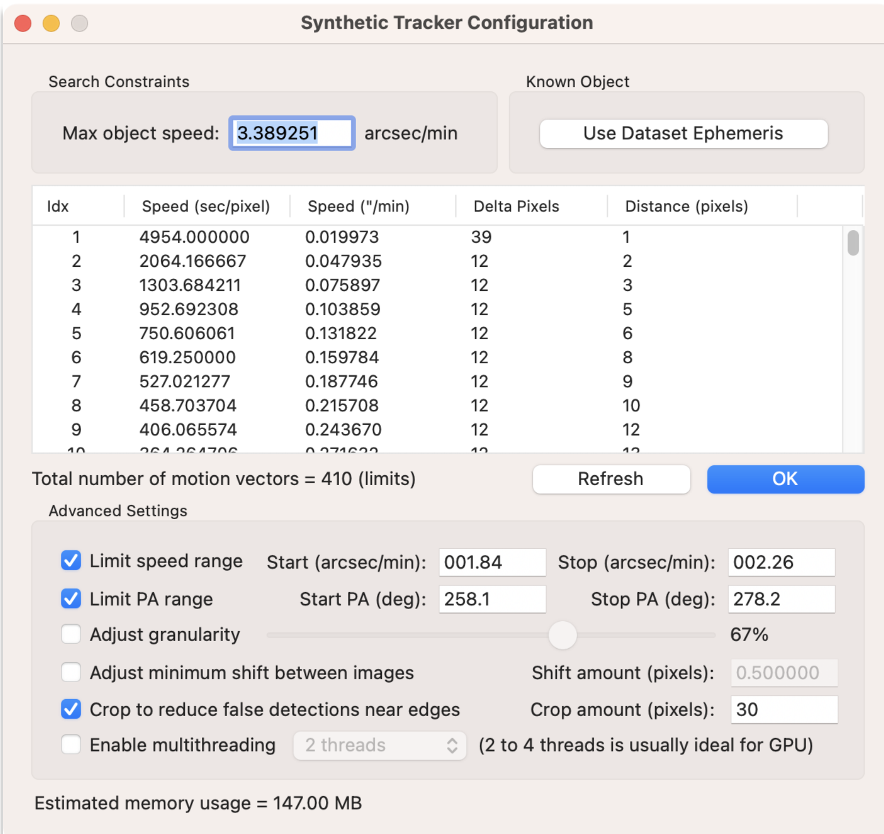


Figure 45 - Synthetic Tracker Configuration

Click "OK" to start the synthetic tracker.

If GPU acceleration is enabled, the tracker should complete after a few seconds; otherwise, it may take a minute or two.

Step 4: Analyze the Tracks

When the tracker has finished, the "Track Navigator" window should appear with a list of candidate detections (tracks). Normally, the target of interest should be in the top 10 tracks, as they are sorted by "quality" by default. In this case, it is track #1. If a NEO you are searching for does not appear in the top ten tracks, you can search for it in the list by going to ***Find Object->By Ephemeris*** from the "Track Navigator" menu and it will sort the tracks by proximity to the expected location of the object as computed by its ephemeris. Alternatively, you could also invoke ***Find Object->RA/Dec*** and input the coordinates manually. In either case, the tracks will be sorted by proximity to an expected location. Note however that just because a track appears to be close to where the object is expected, does not mean that it is in fact the object. You will have to perform either visual inspection or generate measurements that can be validated against other observations.

As the NEO was found in track #1, double-click track #1 to bring up the “Verify Track” window. You should see the motion of the object in the “Image Viewer”. At this point, you can proceed to generate observations, validate those observations, and generate a report, as described in previous examples.

Step 5: Using Orbital Elements to find the NEO

The first example described how to use the “Known Objects” window to locate objects based on their orbital elements. This makes use of the “MPCORB.DAT” file published by the Minor Planet Center.

Another way to find the NEO is to compute its orbital elements from the published observations, and then add those elements to the “MPCORB.CUSTOM” file, which is included in the regular search of the other asteroids.

To do this, either download observations of the object or copy and paste observations from another source into the “Text Form – Observations” window. If you want to download the observations, navigate to ***Tools->Download Observations*** from the main menu, specify “2018 RB” (without the quotes), specify “From obs database (confirmed object)”, and click “OK”.

Once the “Text Form – Observations” window has been populated with observations of the object, you can then generate the orbital elements of the object and store them into the “MPCORB.CUSTOM” file by invoking ***Orbital Elements->Add to Custom Elements*** from the “Text Form – Observations” window. This invokes the orbit computation process to compute the orbit, and after it has finished a new window will appear showing the objects stored in “MPCORB.CUSTOM”. You can then save the result by going to *File->Save* from the menu of this new window. It may present a message saying that the application must be restarted for changes to take effect; however, a shortcut around this is to go to ***Settings->Known Objects*** from the main menu and click the button labeled “Verify Database”, which will force a reload of the “Known Objects” database. The reason this is not done automatically is because if you had previously loaded the database to perform matching with tracks in the “Track Navigator”, those matches will become invalidated upon a database reload.

Having updated the “MPCORB.CUSTOM” file, you can now go to ***File->Load Known Objects*** from the “Image Viewer”. You should now see two instances of the object shown in the “Known Objects” window. The first instance is from the official “MPCORB.DAT” file, and the second instance (at the bottom of the list) is from the modified “MPCORB.CUSTOM” file.

At this point, you can click on the entry in the “Known Objects” window that pertains to the object and a stack will automatically be created (assuming “Auto Stack on Highlight” is enabled from the “Stack” menu). From this stack, you can then double-click on the object to center it in the crosshairs. Then, right-click inside the “Image Viewer” and choose “Create Track – From Current Position”. This will add a new track to the “Track Navigator” window. As before, you can proceed to generate observations using this track information.

If you want to remove the item from “MPCORB.CUSTOM”, go back to the “Text Form – Observations” window and choose ***Orbital Elements->Edit Custom Elements***. Make the necessary changes to remove the item. Then save the changes by going to ***File->Save***.

Step 6: Adjusting the Search Parameters

By now you have learned how to find the NEO using several methods:

1) Creating a Stack (using ephemeris information)
2) Using the Synthetic Tracker (with ephemeris information)
3) Using orbital elements (using known observations)

All of the above approaches require some knowledge of the motion of the object. However, the synthetic tracker can be used to find an object, even if its motion has large uncertainty.

To try this, navigate back to ***Action->Synthetic Tracker*** and specify the same sensitivity threshold as you did earlier. Then, on the last page concerning the "Synthetic Tracker Configuration", try expanding the search as follows:

```
Limit speed range: 1.50"/min to 2.50"/min
   Limit PA range: 200 to 300 degrees
```

This should result in approximately 4500 motion vectors.

NOTE: If you specify an upper bound on speed that is larger than the "Max object speed" constraint, you will need to increase the "Max object speed" accordingly (using "Dataset ephemeris" automatically raises the "max object speed" constraint when necessary, but manual input does not).

Click "OK" to start the synthetic tracker with this expanded search. It may take several minutes, depending on your hardware configuration. On a MacBook with M1 Max, it took 23 seconds.

When the tracker has finished, you will see the results shown in the "Track Navigator". The object should still be shown as track #1 as there are no other objects in this field with similar motion. As the results indicate, the tracker was able to find the object even with a much wider search range. This can be useful when the orbit of a NEO has very few observations, as you can open up the search parameters to account for the increased uncertainty.

Example #6: Discovering New Asteroids with Synthetic Tracker

This example demonstrates the primary use case of synthetic tracking, which is to discover new asteroids, comets, and other objects that are much fainter than could be detected with the conventional technique.

Step 1: Load and Process Images

This example uses the same dataset as the previous example, dataset "ds2". Proceed to load and process the images, applying the steps for calibration, plate solving, and alignment. Note that the option "Fix Hot Pixels" can be a very useful calibration setting, so it is recommended to enable it here. If you need a refresher on how to perform the processing steps, please refer to the first example.

Step 2: Run the Synthetic Tracker

Once the images have been processed, navigate to ***Action->Synthetic Tracker*** from the main menu. The window will prompt for a "Detection Sensitivity Threshold". The setting you choose here depends on how sensitive you want the tracker to be. A setting of 100% is the most sensitive and will generate a large number of detections. Conversely, a setting of 0% will generate almost no detections at all, save for the very brightest objects. It is usually a good idea to allow some false detections to be generated, because that ensures you are detecting at a level that is sensitive enough to detect even the faintest objects. To follow along with this example, use the setting of 50% and click "OK" to continue.

Finally, you will be shown the "Synthetic Tracker Configuration" window.

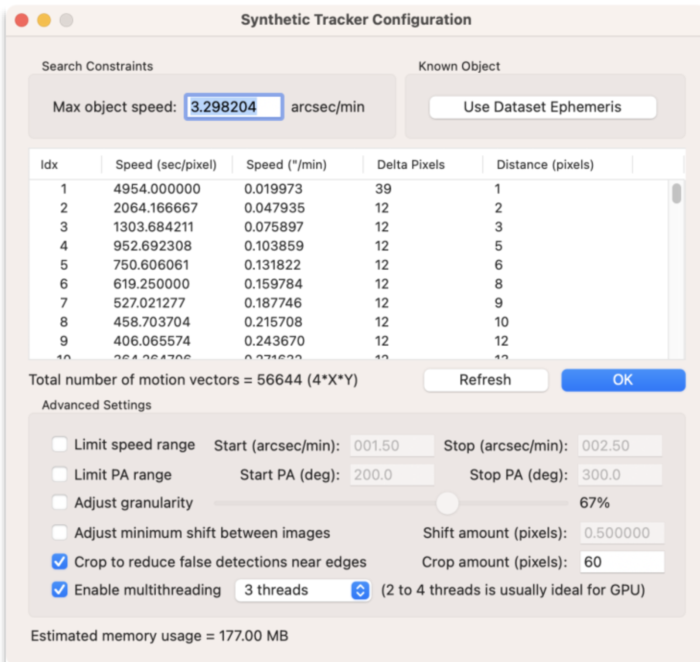


Figure 46 - Synthetic Tracker Configuration

The first parameter, "Max object speed", determines the overall constraint on the search space. By default, it is computed by assuming an object could move as fast as four pixels per exposure (a slight amount of streaking). In other words, if you had a 120 second exposure, and a plate scale of 1.65"/pixel, it would set the default max speed to [1.65*60*(4/120)] = 3.3 arcsec/min. In general, the default setting is usually adequate, but if you have a very fast object (or a very long exposure time), then you may need to adjust this setting.

Under the "Advanced Settings" parameters you will find the ability to limit the search based on speed and position angle. This is very useful if you already know the motion of the object. You can also adjust the granularity of the motion vectors, with higher granularity resulting in more vectors to search. A setting of 67% for granularity is almost always sufficient. The "minimum shift between images" determines the threshold at which a shift is considered significant to warrant inclusion in the list of search vectors. It acts as another level of granularity, with smaller values resulting in more vectors. Usually, the default value of 0.5 is sufficient. Next, you have the ability to crop detection near the edges, a value of 30 to 60 is usually ideal.

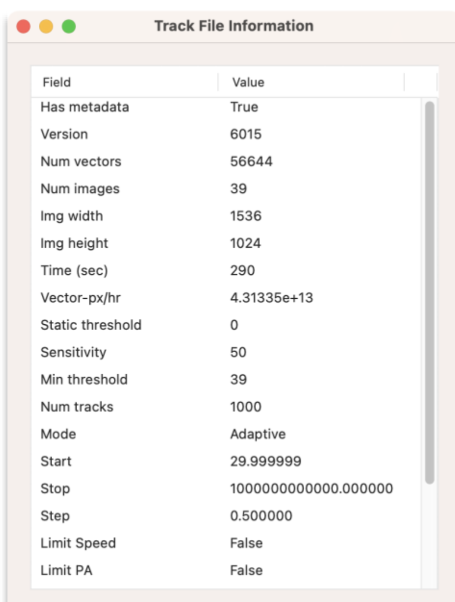
Finally, you may want to enable multithreading to speed up the processing. In CPU mode, this setting overrides the setting specified in “GPU Acceleration” settings, and the threading works by dividing up each image for processing. In GPU mode, this setting works by having each thread operate on its own motion vector, and consequently there is some memory overhead as each thread allocates its own workspace. In general, when in CPU mode, the number of threads should be set to the number of cores, whereas in GPU mode the number of threads should be just enough to keep the GPU busy, which may be only 3 or 4 threads even on a large-core system. This is where you will have to experiment to identify optimal settings for your particular system.

Make sure that your settings match those in Figure 46. Then click “OK” to begin the search.

Because we are doing a “blind” search (meaning no limits on speed or PA), the number of motion vectors is rather high at 56,644. Consequently, this dataset may take some time to process, particularly if GPU acceleration is not enabled. On a MacBook with M1 Max, the total time to process was 290 seconds (less than five minutes).

You can see the performance of your system by going to ***Metadata->View*** in the “Track Navigator” window that appears after the processing has completed.

It is recommended that before you do anything else, go ahead and save the results by going **to *File->Save Tracks*** in the “Track Navigator” window. This way, you can easily revert back without having to run the tracker again.



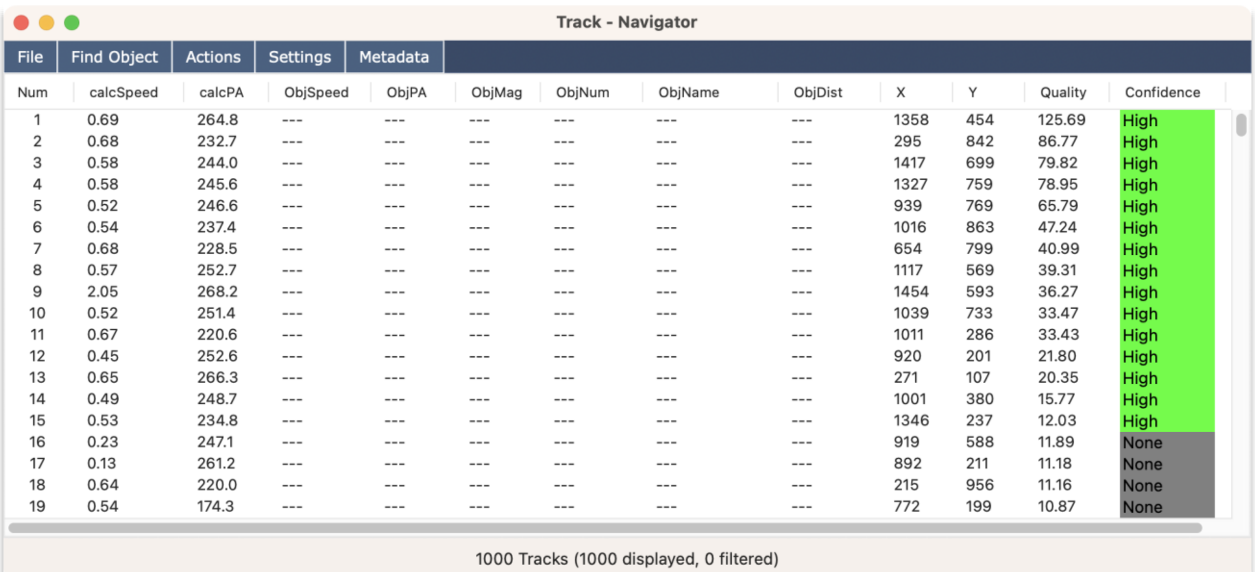
Field	Value
Has metadata	True
Version	6015
Num vectors	56644
Num images	39
Img width	1536
Img height	1024
Time (sec)	290
Vector-px/hr	4.31335e+13
Static threshold	0
Sensitivity	50
Min threshold	39
Num tracks	1000
Mode	Adaptive
Start	29.999999
Stop	1000000000000.000000
Step	0.500000
Limit Speed	False
Limit PA	False

Figure 47 - Track File Metadata

You will note that 1000 tracks were identified, as shown at the bottom of the “Track Navigator” window. As you can see, sensitivity of 50% is sufficient to detect the faintest objects. If you were to re-run it with a sensitivity of 0%, you would see that only 13 tracks are returned in total. Tracks at the top of the list are most likely to be valid, while those ranked farther down the list are more likely to be false detections.

Starting with v6.1, there is now an option to compute a "confidence" metric for each track, making it much easier to determine which tracks are true detections. Navigate to ***Actions->Compute Confidence*** from the "Track Navigator" menu to try out the feature. You can click "Cancel" at any time. The confidence algorithm is very good at distinguishing real objects from random noise. However, camera artifacts such as hot pixels can still appear as "real" objects due to their motion, so you will want to ensure that you have removed such artifacts from the images during calibration.

Tip: If you still want to have high sensitivity, but do not want thousands of tracks returned, go to ***Settings->Tracker*** from the main menu and specify the maximum number of tracks to be returned.



Num	calcSpeed	calcPA	ObjSpeed	ObjPA	ObjMag	ObjNum	ObjName	ObjDist	X	Y	Quality	Confidence
1	0.69	264.8	---	---	---	---	---	---	1358	454	125.69	High
2	0.68	232.7	---	---	---	---	---	---	295	842	86.77	High
3	0.58	244.0	---	---	---	---	---	---	1417	699	79.82	High
4	0.58	245.6	---	---	---	---	---	---	1327	759	78.95	High
5	0.52	246.6	---	---	---	---	---	---	939	769	65.79	High
6	0.54	237.4	---	---	---	---	---	---	1016	863	47.24	High
7	0.68	228.5	---	---	---	---	---	---	654	799	40.99	High
8	0.57	252.7	---	---	---	---	---	---	1117	569	39.31	High
9	2.05	268.2	---	---	---	---	---	---	1454	593	36.27	High
10	0.52	251.4	---	---	---	---	---	---	1039	733	33.47	High
11	0.67	220.6	---	---	---	---	---	---	1011	286	33.43	High
12	0.45	252.6	---	---	---	---	---	---	920	201	21.80	High
13	0.65	266.3	---	---	---	---	---	---	271	107	20.35	High
14	0.49	248.7	---	---	---	---	---	---	1001	380	15.77	High
15	0.53	234.8	---	---	---	---	---	---	1346	237	12.03	High
16	0.23	247.1	---	---	---	---	---	---	919	588	11.89	None
17	0.13	261.2	---	---	---	---	---	---	892	211	11.18	None
18	0.64	220.0	---	---	---	---	---	---	215	956	11.16	None
19	0.54	174.3	---	---	---	---	---	---	772	199	10.87	None

Figure 48 - Track Results

Step 3: Analyzing the Tracks

At this point you have a list of numerous tracks available to evaluate. It would be quite tedious to examine them all in detail, so one task to carry out is to determine which tasks match up with an already known object. To do this, go to ***File->Load Known Objects*** from the "Track Navigator". You will now see that most of the tracks in the first 20 entries are matched with a known object. One of the tracks -- number 17 -- is an erroneous match, having an "ObjDist" value exceeding 0.5 arcminute, while the other matches have an "ObjDist" value under 0.1 arcminutes, indicating a good match. Again, this is a configurable setting: you can set the match tolerance such that erroneous matches are less likely to appear. However, some objects have higher uncertainty in their orbital elements, and so a higher tolerance is desired. Finally, track #15 appears to be a potential discovery, as it does not match up with any known object. So, in this scenario, you would proceed to create observations of that object by double-clicking on its track entry (or right-clicking on it, and choosing "Verify Track" from the popup menu that appears). With the "Verify Track" window open, you should now see an animation of the track. You can visually inspect the object by examining the animation in the "Image Viewer".

Tip: The animation is one way to verify that the object is real. Another way is to scroll through the images in the "Track – Positions" window, which will present each image in the "Image Viewer" as you scroll down using the "Down" arrow key. From this dialog you can also use the left and right arrow keys to navigate to other tracks, which is much more convenient than going back and forth to the “Track Navigator” window.

Proceed to create observations of this object by clicking the “Add Observations” button.

Num	calcSpeed	calcPA	ObjSpeed	ObjPA	ObjMag	ObjNum	ObjName	ObjDist	X	Y	Quality	Confidence
1	0.69	264.8	0.69	264.8	18.1	(38826)	2000 RZ92	0.016	1358	454	125.69	High
2	0.68	232.7	0.68	232.5	18.8	(128657)	2004 RN50	0.026	295	842	86.77	High
3	0.58	244.0	0.58	243.6	18.8	(213074)	1999 TH90	0.053	1417	699	79.82	High
4	0.58	245.6	0.59	245.5	18.9	(81801)	2000 KU3	0.033	1327	759	78.95	High
5	0.52	246.6	0.53	246.1	19.3	(63343)	2001 FO85	0.030	939	769	65.79	High
6	0.54	237.4	0.54	237.6	19.4		2014 TH28	0.047	1016	863	47.24	High
7	0.68	228.5	0.68	227.8	20.7		2016 AS350	0.054	654	799	40.99	High
8	0.57	252.7	0.57	252.4	19.8	(318775)	2005 SF90	0.024	1117	569	39.31	High
9	2.05	268.2	2.06	268.1	19.8		2018 RB	0.020	1454	593	36.27	High
10	0.52	251.4	0.52	251.2	20.3	(97542)	2000 DD43	0.023	1039	733	33.47	High
11	0.67	220.6	0.67	220.5	20.0		2018 RX19	0.019	1011	286	33.43	High
12	0.45	252.6	0.45	253.2	20.6		2018 RW19	0.037	920	201	21.80	High
13	0.65	266.3	0.64	265.6	20.5		2000 EE117	0.019	271	107	20.35	High
14	0.49	248.7	0.50	249.4	20.8		2005 EQ301	0.042	1001	380	15.77	High
15	0.53	234.8	---	---	---	---	---	---	1346	237	12.03	High
16	0.23	247.1	---	---	---	---	---	---	919	588	11.89	None
17	0.13	261.2	0.45	253.2	20.6		2018 RW19	0.846	892	211	11.18	None
18	0.64	220.0	---	---	---	---	---	---	215	956	11.16	None
19	0.54	174.3	---	---	---	---	---	---	772	199	10.87	None

Figure 49 - Matching Tracks with Known Objects

As this is an unknown object, leave both the permanent ID and the provisional ID fields blank. Only the “Tracklet ID” field should be populated, with an identifier of your choice (however, the computer-generated identifier should normally work fine).

Step 4: Using MPChecker to Determine if Object is “New”

At this point, you have created observations of a potential discovery. Now you will want to use the “MPChecker” tool to see if these observations might still match up with an already known object. To do this, navigate to the following webpage and input the observations of the object into the web form:

https://www.minorplanetcenter.net/cgi-bin/checkmp.cgi

Click the radio button “these observations” and then click “Produce List”. If the results come back with “No Known Objects”, or if there are objects but the offsets are all greater than 0.1, then it may well be a discovery. Note that the MPChecker tool requires observations in MPC1992 format.

Step 5: Taking Optimal Data for the Tracker

As you look through the tracks in the list, you can see that the synthetic tracker does quite well at detecting very faint asteroids. It even detected “2018 RB” in a fully blind fashion. However, there are some limitations of the tracker to be aware of so that you can better understand how to optimize the data for it.

For one, if your data has numerous “hot pixels” or other artifacts, then the tracker will produce subpar results. This is because, although hot pixels may appear stationary on the un-aligned images, once the images have been aligned, the hot pixels (and other artifacts) can now appear to have motion from one image to the next. So, for best results, use a good dark frame, or some other mechanism to suppress the hot pixels and other camera artifacts. In this example, the “Fix hot pixels” calibration option was used, which can be helpful at further mitigating hot pixels. It is also recommended to perform dithering between exposures as doing so will reduce correlated noise artifacts.

Second, be sure to take data that matches the motion of the object you wish to detect. If it is a very fast-moving object, use short exposures of perhaps 10 seconds or even shorter. Otherwise, if the object is slow-moving, you could use longer exposures such as 30 or 60 seconds. But most importantly, be sure that the *total exposure time* is long enough so that the object exhibits sufficient motion. That is to say, you could get away with using 10 second exposures even on a very slow-moving object: so long as the total exposure time is long enough for the object to move at least 5 pixels, then the object should be detectable. Generally, the first constraint for the tracker is the number of images, with a minimum of 11 required. And in fact, it is best to have at least 25 images, preferably 30-60 images. With CMOS cameras having low readout noise, you can usually get away with shorter exposures, and simply have more of them to reach the desired exposure count. If you are unsure of what exposure times to use for a given object, navigate to ***Calculators->Optimal Exposure Time*** from the main menu.

Example #7: Longer Total Exposure Dataset (with Synthetic Tracking)

This example is very similar to the previous example, except that it has a longer total exposure time using 60 images.

Step 1: Load and Process Images

Proceed to load and process the images, performing calibration, plate solving, and alignment. The dataset to use is that of “ds3”.

Step 2: Run the Synthetic Tracker

Navigate to ***Action->Synthetic Tracker***. Use sensitivity of 50%. For the tracker configuration, do not apply any limit to speed or PA. There should be 121,104 vectors to be searched. Use crop of 60. Granularity of 67% (default). Then click “OK” to start the tracker. With the total time increased from 1.4 hours to 2.2 hours, the tracker has to search a much larger number of trial vectors (from around 56k vectors to 121k vectors), so the processing time will also be increased. On MacBook with M1 Max, it took 345 seconds to complete.

Step 3: Analyze the Tracks

After the tracker has finished, you will note that it has found a total of 16 asteroids rated with “High” confidence. Your track numbers may be somewhat different if the images were processed with a different calibration and/or alignment settings. Also, just because a track is rated with “High” confidence does not automatically mean that it is a true target, but it does help to narrow down which tracks need to be examined.

As you can see from these results, the additional dwell time and more exposures enabled the tracker to identify more asteroids, including 2008 WY119 which is at magnitude 21. Note however that there are diminishing returns to the longer dwell time. Especially with NEOs, where they will move out of the field of view more quickly and therefore become undetectable, or eventually introduce curvature in their motion. You will have to determine the optimal dwell time for the type of target you wish to detect.

Example #8: Using the Test Target Generator

A more scientific way to evaluate the performance of the tracker with respect to different threshold settings is to inject a grid of "test targets" that have a known signal-to-noise ratio (SNR). By running the tracker on identical data but with different threshold settings, it is possible to more easily quantify the detection performance.

Step 1: Load and Process Images

Load the images from "ds2" and proceed to process them by performing calibration, plate solving, and alignment.

Step 2: Run the "Evaluate Thresholds" Module

Navigate to ***Action->Evaluate Thresholds*** from the main menu. Here you will see a new window appear with settings for "FWHM", "SNR", "Sensitivity", and "Granularity", along with a few other options.

Specify the following settings:

FWHM			SNR			Sensitivity			Granularity		
Start	Stop	Step	Start	Stop	Step	Start	Stop	Step	Start	Stop	Step
2.5	2.5	0.5	0.6	0.8	0.1	20	45	5	20	45	5

Also set FWHM to be in pixels, and enable the option "Compute magnitude of injected targets". Then click the "Start" button to start the process.

Step 3: Analyze the Results

After a minute the module will finish. There should be three different magnitudes to examine since three sets of targets were injected (SNR=0.6, 0.7, and 0.8). As you can see, detection of SNR=0.6 targets is quite minimal as these are simply too faint to be detected in just 39 images. Detection of SNR=0.7 targets is improved, and as expected, targets with SNR=0.8 are detected the best.

You can hover the cursor over each grid element to see the exact detection performance for that combination of sensitivity and granularity. At the lower-left corner, detection is worse due to having a low sensitivity (20%) and granularity (also 20%). At the top-right corner, detection is best due to having higher sensitivity (45%) and granularity (also 45%). But the trade-off in increased sensitivity is processing time and number of false detections (which are partly mitigated via the "Compute Confidence" routine).

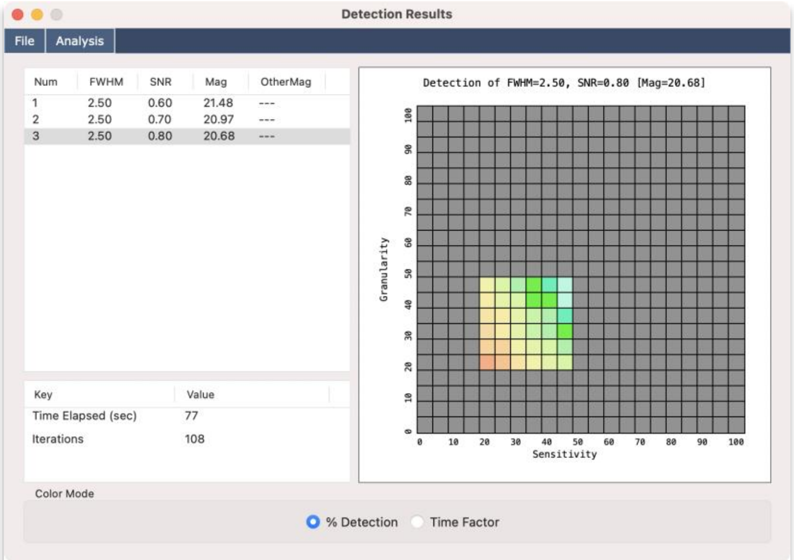


Figure 50 - Detection with 39 Images

Step 4: Compare with only 13 Images

The previous step demonstrated that it is possible to detect 95 out of the 100 injected targets at SNR=0.8, which corresponds to approximately magnitude 20.7 on these images. Now, re-run the evaluation routine again, but this time use only the first 13 images. You should see a noticeable reduction in detection, as shown in Figure 51:

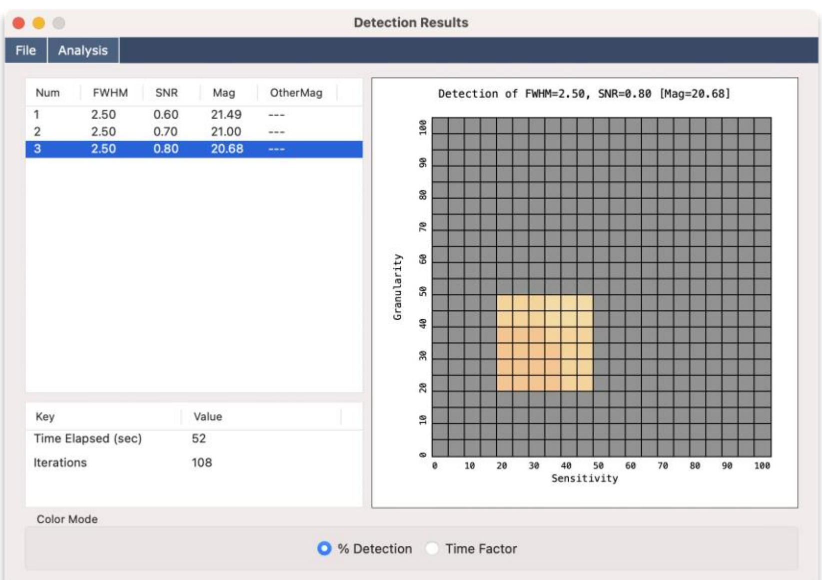


Figure 51 - Detection with Only 13 Images

With only 13 images, detection was reduced to 45 out of the 100 targets, using the same thresholds of sensitivity=45% and granularity=45%. As you can see, there is indeed quite a difference in detection capability with respect to the number of exposures.

Cluster Computing

A single computer (or graphics card) may take a few hours to conduct a full "blind search" of a dataset. If it is desired to expedite the search process, Tycho offers the ability to divide the workload among multiple compute nodes. In this hierarchy there is one primary node and one or more worker nodes. The worker nodes can be configured by installing Tycho on each node and going to ***Network -> Manage Worker Nodes for this Machine*** from the main menu.

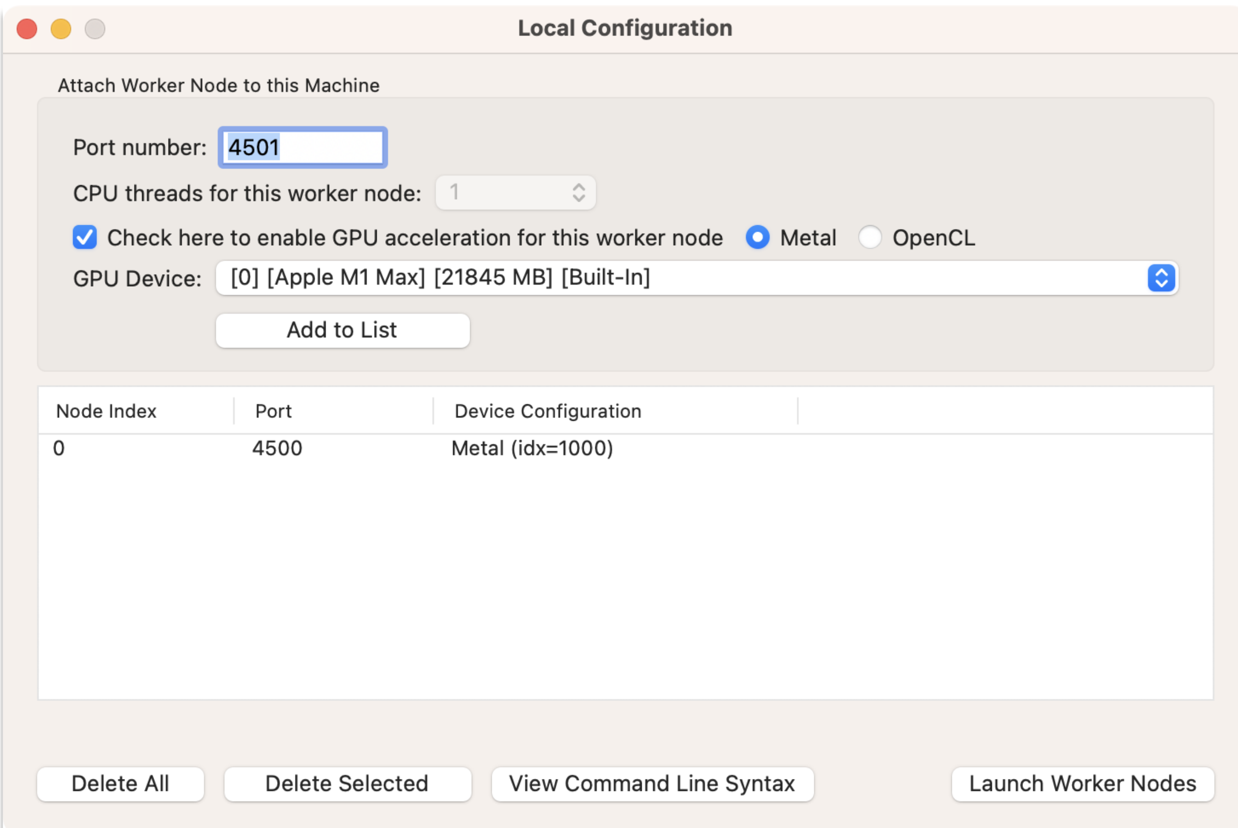


Figure 52 – Local Worker Node Configuration

Figure 52 shows the local configuration for a single machine that has one GPU. Each GPU is assigned a device index and port number. After adding the worker nodes to the list, you can then click the button "View Command Line Syntax" to see an example batch file script that could be used to launch the worker nodes from the command line. Alternatively, you could launch them from this dialog box by clicking the "Launch Worker Nodes" button.

New in v7.4 is the ability to use CPU, rather than GPU, worker nodes. Simply leave the box for GPU acceleration unchecked, and the worker node will instead rely solely on CPU performance. This may be useful for those who do not have access to GPU hardware.

Once you have configured the desired worker nodes, you must instruct the primary node to use them. This is achieved by going to ***Network -> Manage Worker Nodes for Cluster***. Refer to Figure 53.

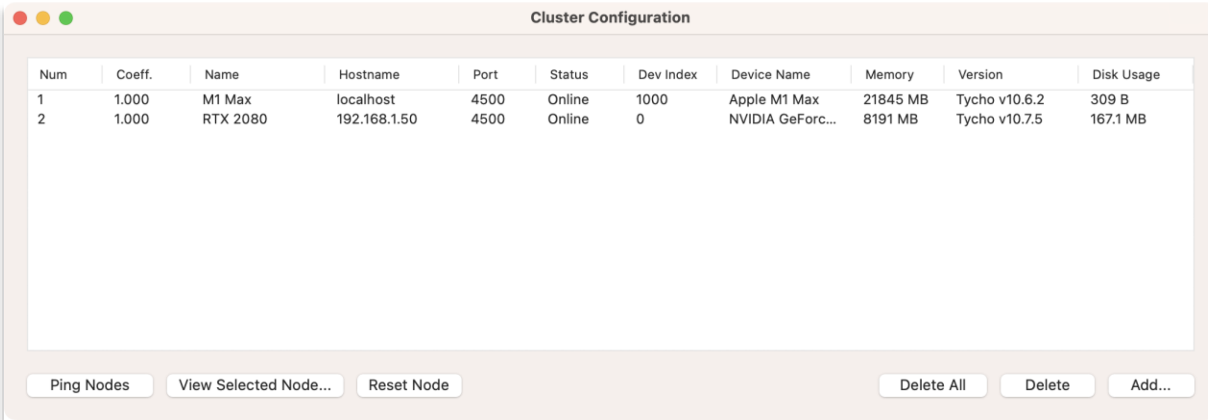


Figure 53 - Managing Worker Nodes

Initially there are no worker nodes. Click the "Add…" button (located in the lower-right corner) to add a new node, and specify the hostname and port number. You should also specify a name for the node. Then click "OK" and you will see the node appear in the list. In Figure 53 you can see that two nodes were added. You can also view information on each node by clicking the "Ping Nodes" button.

If you need to clean up disk usage on a node you can click on "View Selected Node…" and a file browser will be presented. You can delete files and folders here. However, this is generally not necessary as the files are cleaned out at the start of each tracker run.

Having specified at least one worker node, Tycho will now display a cluster configuration screen when you run the synthetic tracker. By default, it will show the primary node (Local machine) and also the worker nodes that you added earlier. If you wish to exclude a node from being used, you may select it and click "Deallocate Selected". This will move the selected node to the list of "Unallocated Nodes". The tracker will only run on the nodes shown in the "Allocated Nodes" list. For example, if you deallocate the primary (local machine) node, then the tracker will only run on the worker nodes.

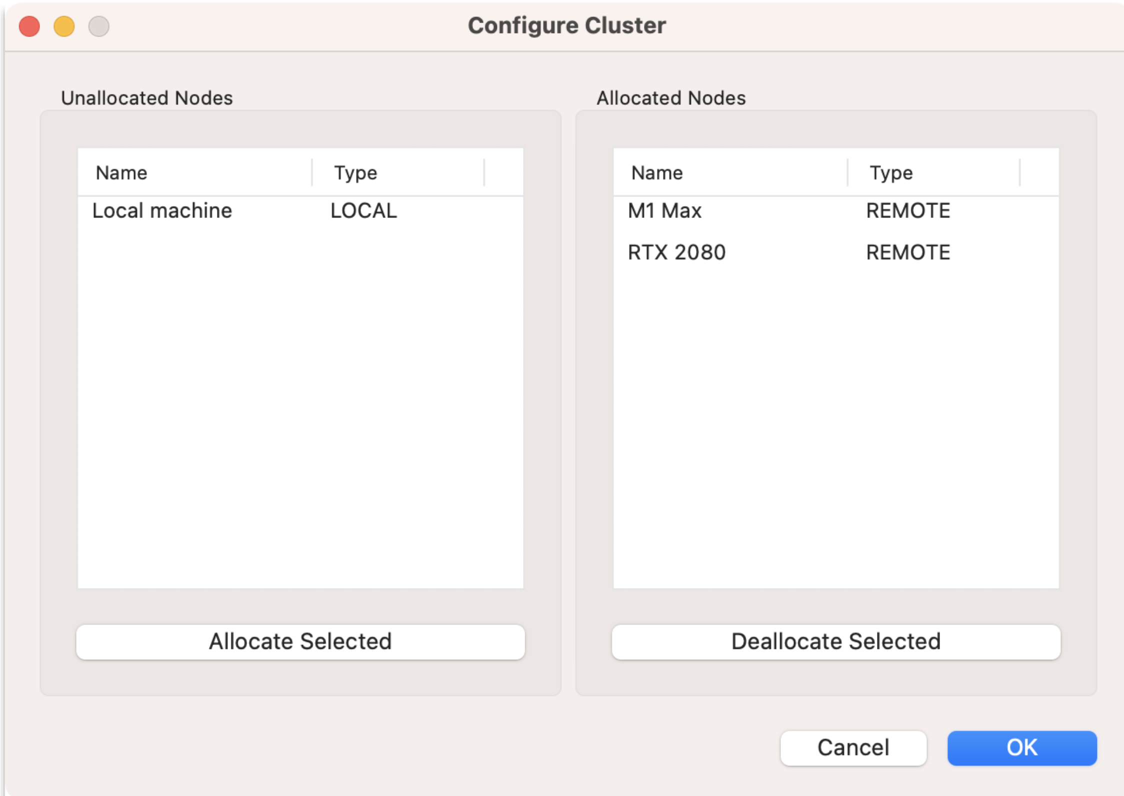


Figure 54 - Cluster Configuration

Once you have configured the cluster as desired, click the “OK” button to proceed.

The next screen that appears is the “Cluster Status” screen, refer to Figure 55. On this screen you are presented with the list of nodes on which the tracker will run (the allocated nodes from earlier). Be sure that all nodes have returned a status of “Ready”, and then click the “Start” button to initiate the cluster processing routine.

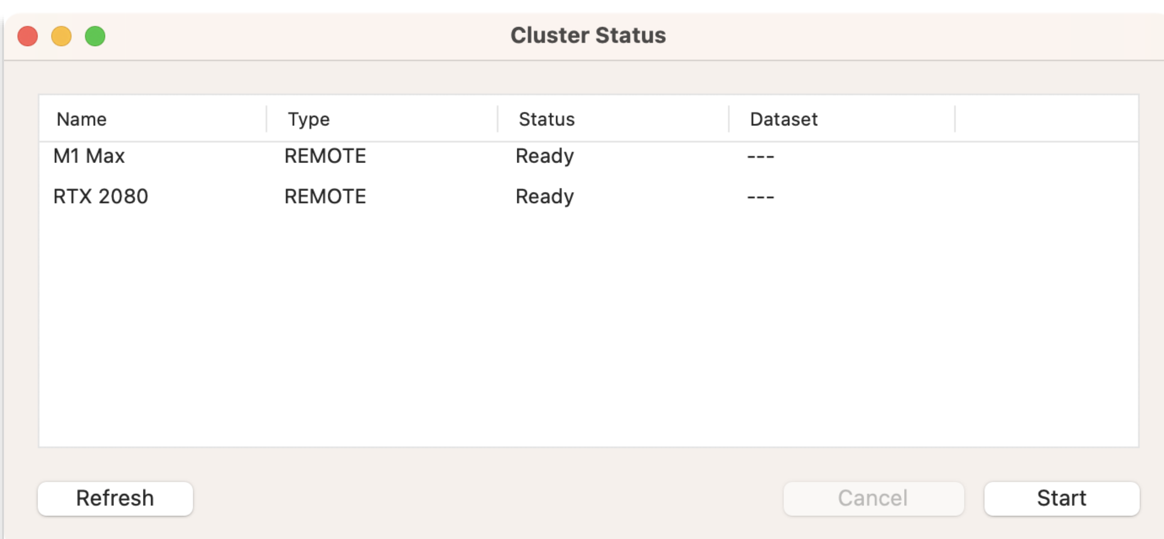


Figure 55 - Cluster Status

As each node processes the data, you will see the status of each node shown in the “Status” column. Refer to Figure 56.

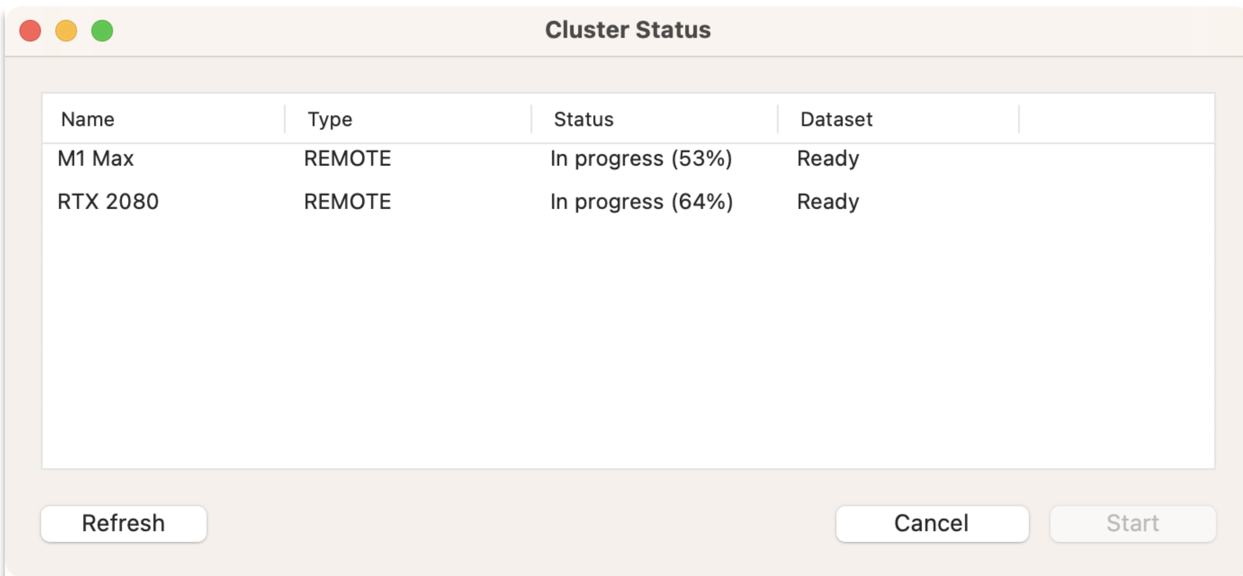


Figure 56 - Node Progress

In this example, the first node is running on a MacBook with M1 Max, and the second is a workstation with an NVIDIA RTX 2080 GPU. As can be seen from the “Status” column, the RTX 2080 is marginally faster than an M1 Max. Consequently, the cluster processing can be optimized further through the use of workload coefficients. These coefficients specify how much work to allocate to the different processing nodes. The “Local” node always has a coefficient of 1.0. Thus, if you add a remote node that is twice as fast, it should be specified as having a workload coefficient of 2.0. Alternatively, you could remove the local node and use only remote nodes, which would allow for an arbitrary coefficient scheme to be used. For example, if there is a worker node having a coefficient of 100.0 and another having a coefficient of 200.0, the work will be divided up so that the former has 1/3 of the work and the latter receives 2/3 of the work.

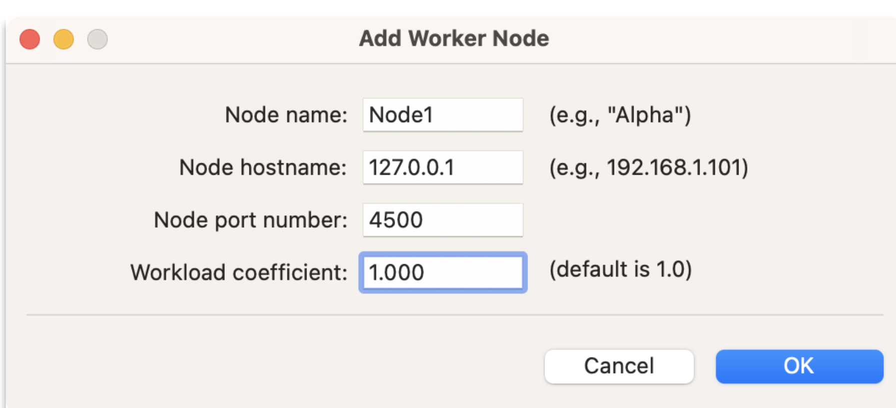


Figure 57 - Adding a Node with Workload Coefficient

Express Mode

Tycho can also combine the processing steps (calibration, plate solving, and alignment) into a single step through the Express Mode feature.

Check the box next to each step to indicate whether or not to perform that particular step. Settings for each step are applied by clicking the "Settings…" button next to the step and specifying the relevant configuration. See Figure 58 for details.

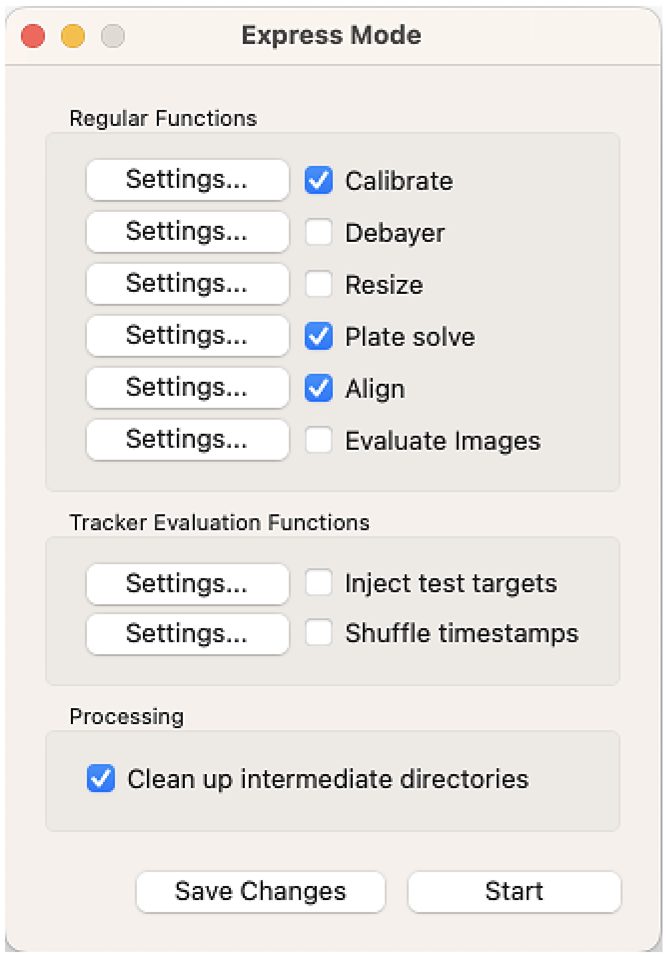


Figure 58 - Express Mode

Auto Run (Scripting)

Tycho offers the ability to process a set of files from the command line or through a batch script. This feature is called "Auto Run" and operates on a set of pre-defined settings which can also be overridden through the use of an "override" file. **Note**: new in Tycho 13 is the ability to use Lua scripts instead of the Auto Run module. This is discussed in more detail in the following section.

In order to use the Auto Run feature, first navigate to ***Settings -> Auto Run*** from the main menu. You will see a new screen appear as shown in Figure 59. The runtime settings determine the behavior of the auto run module, such as whether or not it will perform the steps indicated by "Express Mode". You can also specify the contrast and intensity levels for the generated snapshot images. The "Advanced Settings" are similar to those discussed in the earlier section covering the synthetic tracker.

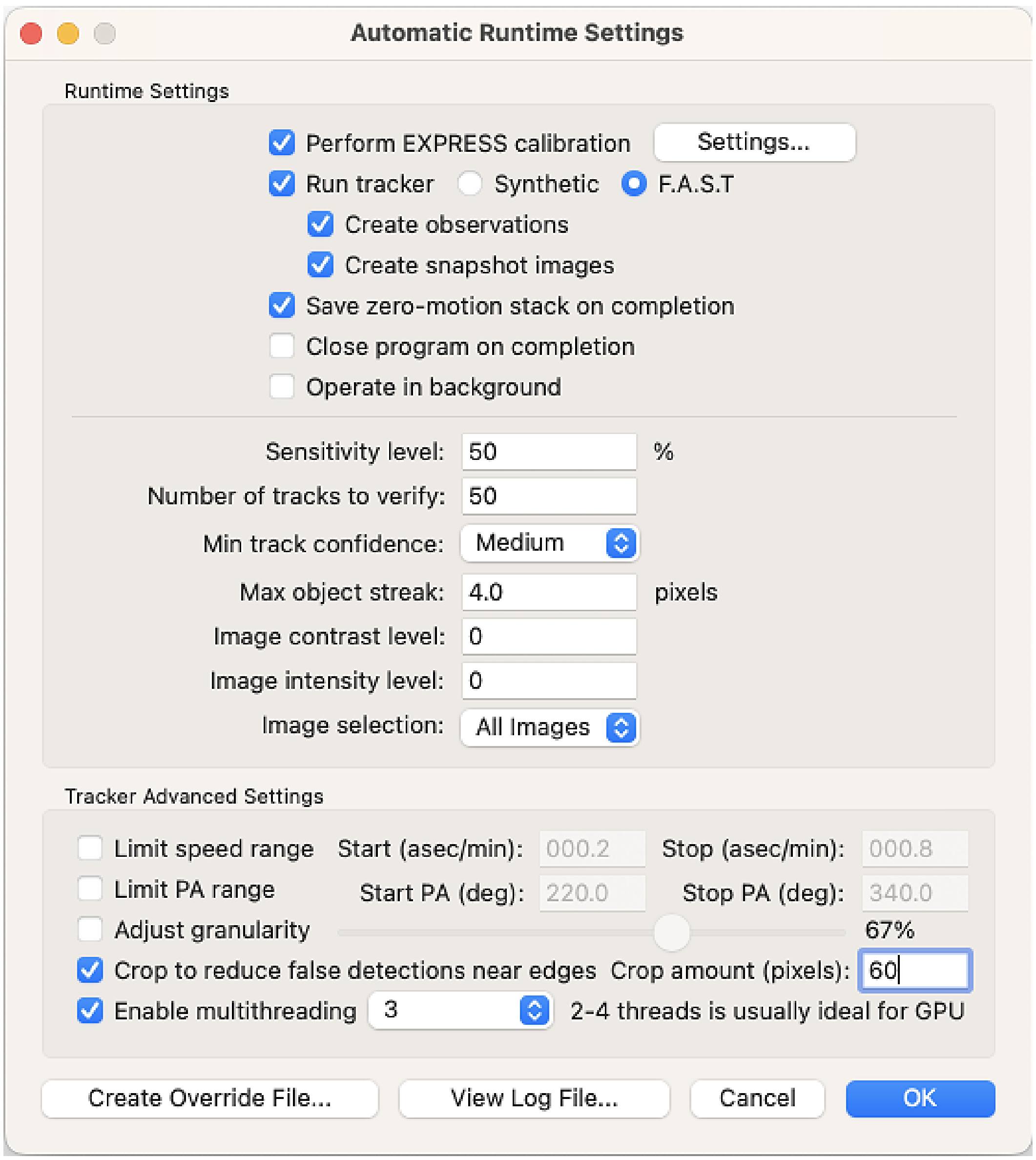


Figure 59 - Auto Run Settings

Once you have configured the desired settings for Auto Run, click "OK" to save the settings. Then, you can open a terminal window and issue a command to Tycho in the following format:

```
Usage: <tycho> <mode> <image directory> [override file]
                    <tycho> = tycho executable
                     <mode> = 1
          <image directory> = path to input directory
            [override file] = path to override file (optional)
```

As an example, for Windows:
"C:\Program Files\Tycho\Tycho.exe" 1 "C:\Users\Daniel\Desktop\iTelescope\data\pipeline\r1"

For macOS:
open -n "/Applications/Tycho.app" --args 1 "/Users/dparrott/Desktop/data/ds3"

The above command will launch Tycho in auto run mode and process the files located in the specified subdirectory.

Another way to use the Auto Run feature is to create a batch file, such as "auto_run.bat" (or "auto_run.sh" for macOS), which provides the path to Tycho and a default image directory. One might then simply replace this directory whenever a new set of images are to be processed.

Finally, there is also the ability to create and use an override file. An override file will, as its name implies, override the settings that you specified earlier. To create an override file, you can click "Create Override File…" from the lower left corner of the window as shown in Figure 59. Then, copy and paste the text into a new text file and save the file as "override.txt" adjacent to the input directory of images. In other words, the override file should be placed in the same directory as the image directory, not inside the image directory itself. In this fashion, you can then write a script to update the override file as desired and the tracker will run accordingly.

Starting with version 6.2, it is now possible to specify the path to the override file as the last argument in the command line invocation.

New in v11.1 is the ability to supply an observation file of a specific object. This observation file is then used by AutoRun to automatically apply motion limits according to the derived ephemeris. Additionally, a motion offset file can be supplied to further refine the search parameters.

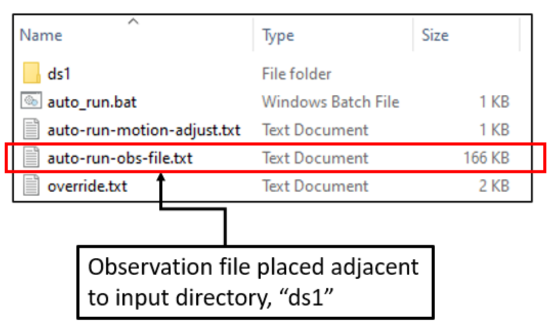


Figure 60 - Supplying Observation and Motion Offset Files for AutoRun

The optional file “auto-run-motion-adjust.txt” specifies the offsets to the motion limits, and is formatted as follows:
SPEED_START=<offset to object initial search speed, in arcseconds/minute>
SPEED_STOP=<offset to the object final search speed, in arcseconds/minute>
PA_START=<offset to the object initial search position angle, in degrees>
PA_STOP=<offset to the object final search position angle, in degrees>
The optional file “auto-run-obs-file.txt” specifies observations (measurements) of a particular object, which AutoRun will then attach as ephemeris to the dataset. This ephemeris will then narrow the search to the computed motion bounds of the object plus the offsets (if any) specified by the “auto-run-motion-adjust.txt” file.

Lua Plugins (Scripting)

Version 13.0 of Tycho introduces the ability to perform various functions through a Lua script. In fact, all of the Auto Run functionality can now be replaced through Lua scripts.

There are two ways to invoke a Lua script in Tycho:
(1) Through the Plugins menu
(2) Through the command line

Plugins menu:
If the Lua script (plugin) has not yet been installed:
1. Navigate to ***Plugins->Plugin Manager*** from the main menu.
2. Choose ***Action->Install Local Plugin*** from the menu of the “Plugin Manager” window.
3. Point to the manifest.txt file associated with the plugin.

If you are new to plugins, it may be worthwhile to download the example plugin “P1 Example Plugin”, which provides a good starting point and a basis for developing new Lua plugins:
1. Navigate to ***Plugins->Plugin Manager*** from the main menu.
2. Choose ***Action->Download Plugin*** from the menu of the “Plugin Manager” window.
3. Right-click “P1 Example Plugin” and choose Install from the pop-up menu.
4. Restart Tycho

Viewing the contents of a plugin:
1. Navigate to ***Plugins->Plugin Manager*** from the main menu.
2. Right-click on the plugin and choose “View in Explorer” (or “View in Finder” on macOS)

Command Line:
Lua scripts can also be invoked via the command-line. Refer to the section “Command Line Interface” for details on the syntax.

Lua API Reference

The following document provides the list of available host functions that can be called from a Lua script:

https://downloads.tycho-files.com/docs/plugin_api.html

Pseudo-Flat Calibration

Version 3.0 of Tycho introduces a way to process raw images with a 'pseudo-flat' technique. The resulting images are both flat and also normalized with a consistent background level. In other words, if you perform pseudo-flat calibration, then you will not need to perform the 'normalization' step.

To perform this calibration, go to ***Action->Calibrate Images***. For "Flat Frame", specify "Use pseudo flat", and then click the "Settings…" button to adjust the settings as desired (usually the defaults are ideal).

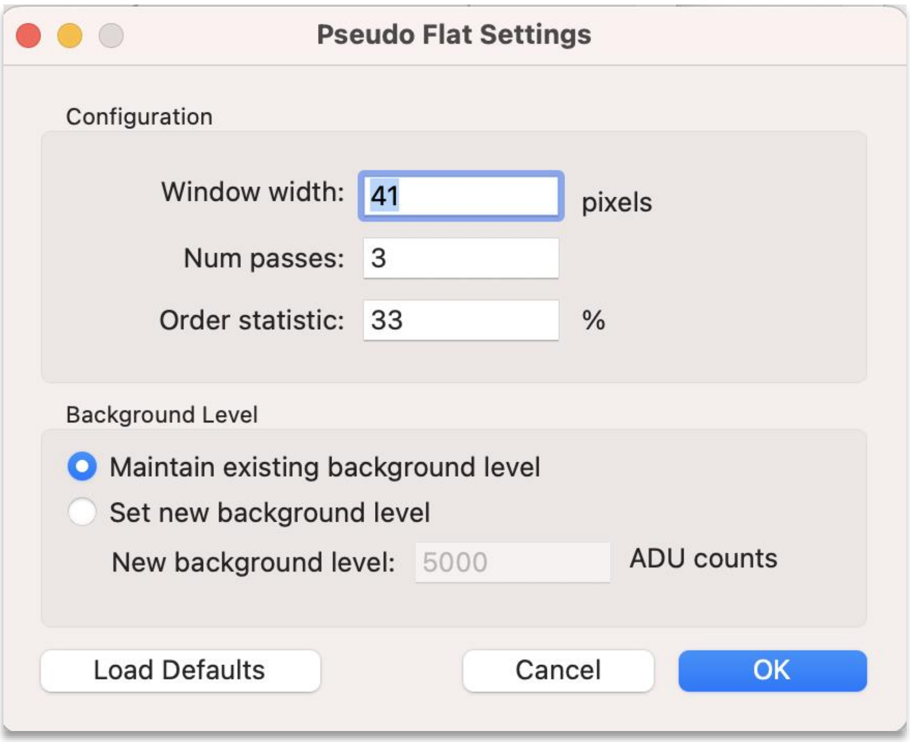


Figure 61 - Image Calibration

By default, a window width of 41, with three passes (3), and an order statistic of 33% is ideal. You may also specify a new background level or maintain the existing background level.

Dataset Ephemeris

Another new feature introduced in v3.0 is the ability to directly attach object ephemeris to each image. This can be useful for those performing NEO confirmation, as an initial starting point in determining what motion vector(s) to use in finding the object.

First, go to ***Tools->Download Observations*** from the main menu. Then type in the object name, and choose whether or not to retrieve the data from the NEOCP list or from the Observations database:

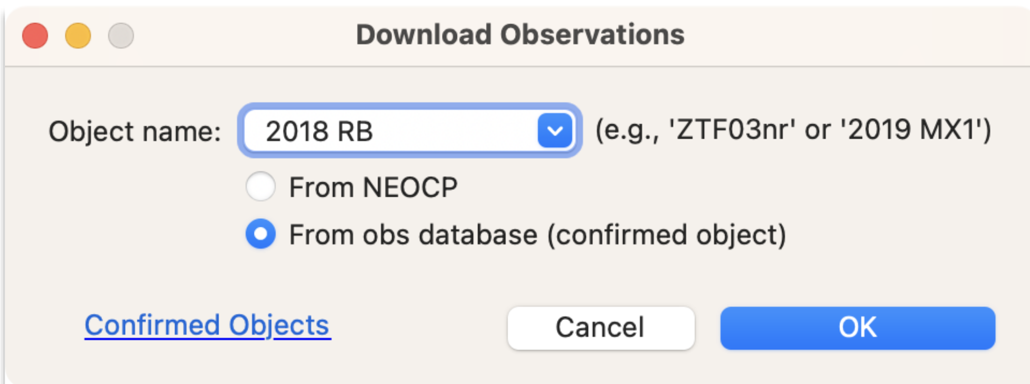


Figure 62 - Download Observations

If the object has been confirmed, then it will no longer be on the NEOCP list, and you will have to choose "From obs database (confirmed object)".

Now click "OK" and the observations of the object will be shown (it may take a moment to retrieve the data from the MPC server).

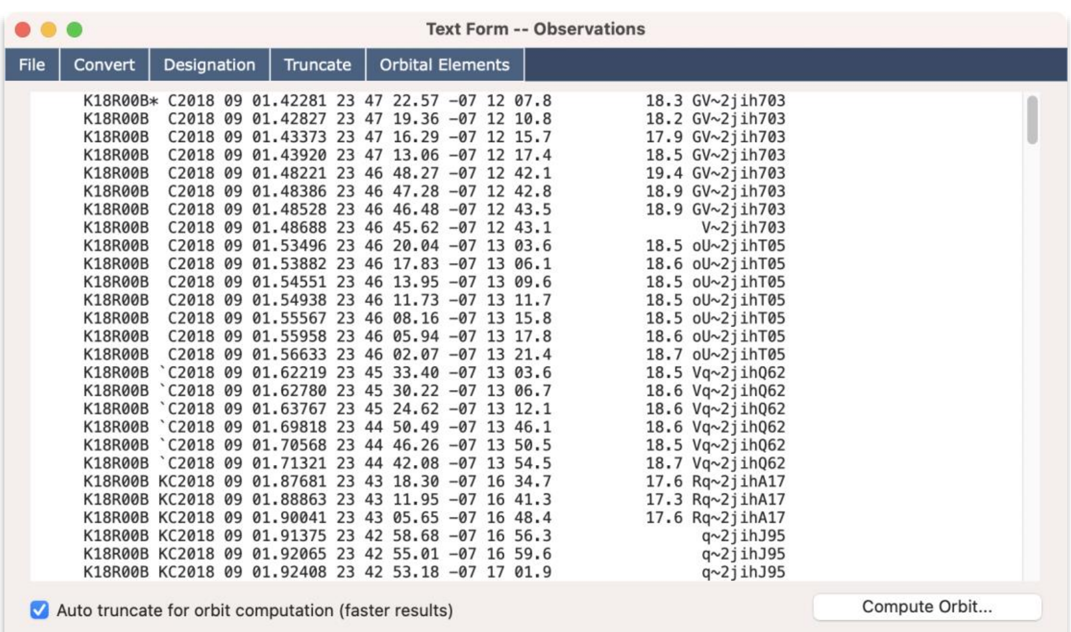


Figure 63 - Downloaded Observations

At this point, you can click “Compute Orbit”, and an orbit will be computed from these measurements. Once the computation has completed, a new orbit window will appear. From this new window (titled “Orbit: 2018 RB” for this example), choose ***Ephemeris->Attach to Dataset***. After a moment, the ephemeris information of the object will be applied to each image listed in the “Image Manager”.

At this point, assuming all went well, you should see the columns in the “Image Manager” populated with ephemeris information for each image. Particularly useful is the last column, “EPH_IN_FOV”, which indicates whether or not the object is expected to be within the field of view of the image.

For those images in which the object is not within the field of view, it is advised to remove them from the dataset as they will simply degrade the resulting stacks and measurements. To do so, select the images, and then choose ***List->Remove Selected*** from the “Image Manager” menu.

Another reason why the dataset ephemeris is helpful is that it can be used to provide an initial starting point for conducting the search of an object. For example, in the configuration for the synthetic tracker, you can click “Use Dataset Ephemeris” (located near the top-right), and it will then update the limits for speed and position angle accordingly.

While this will work for 99% of objects, sometimes a very new discovery that has only a few observations may not have a well-defined motion, so be careful when using the dataset ephemeris too strictly. It may be necessary to open the search space, particularly for the speed. For example, on object “2019 RC”, the dataset ephemeris suggested that the object was moving between 4.04”/min and 3.59”/min, when in truth, the correct motion was around 3.18”/min. Had the dataset ephemeris been used directly, the object would not have been found. The solution is to widen the search space for such objects where the orbit is not yet well-known.

Repositories

New in v5.0 is the concept of repositories, which provide a convenient way to store observations. For example, one repository might be associated with 'interesting' objects for further follow-up, while another repository might simply store all observations collected to-date. There are also two repositories created for the purpose of storing "Near-Earth Object Confirmation Page" (NEOCP) objects and the "Recently Confirmed" objects published by the Minor Planet Center (MPC).

One use case for the latter two repositories is to determine if an object you found matches up with an object listed on the NEOCP or the "Recently Confirmed" page. In other words, while the "MPChecker" tool (mentioned earlier) does a good job at identifying known objects from a set of observations, it does not check recently confirmed objects (nor objects on the NEOCP page), so it is helpful to have another tool that can do so. Searching for such a match is achieved through the use of the "Object Linker" tool, accessible via ***Tools->Identify Linkages*** and covered in a later section.

Repositories can be accessed via ***Tools->Manage Repositories*** from the main menu.

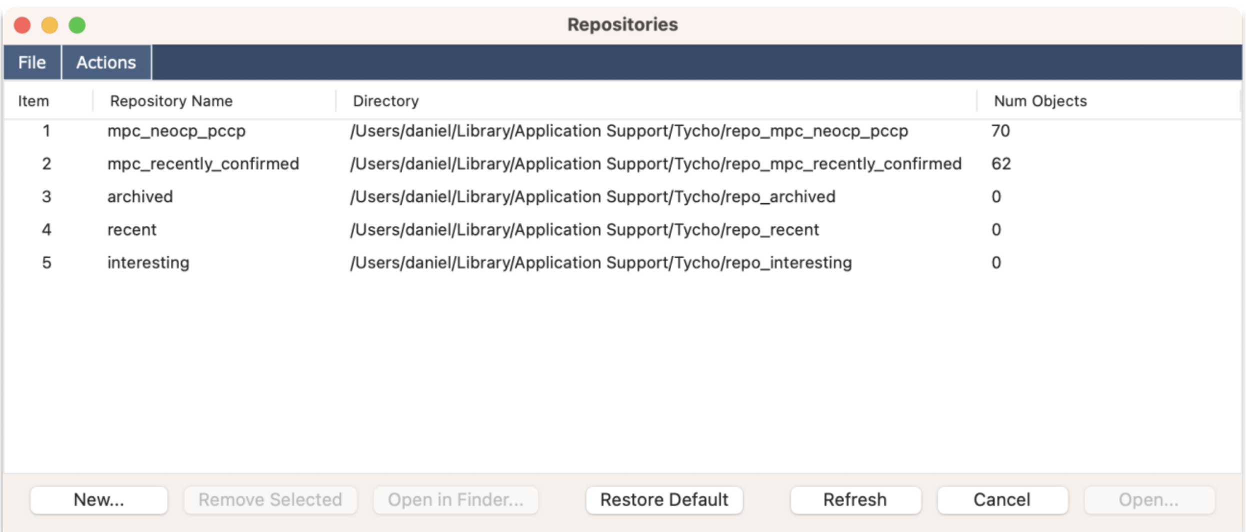


Figure 64 - Manage Repositories

Saving Observations to a Repository

Observations can be saved to the repositories through three different methods. For the "NEOCP/Recently Confirmed" repositories, navigate to ***Actions->Download NEOCP/Recently Confirmed*** from the "Repositories" menu. Another method is through the "Observations -- All Targets" window, where it is possible to save the observations via ***File->Export to Repository***. Doing this will group the observations by object and save each object as a text file to the chosen repository. The third method is through the "Text Form -- Observations" window, where the observations in the edit box can be saved to a repository via ***File->Save to Repository***. In this case, all of the observations in the edit box are saved to a single text file in the chosen repository.

Clearing a Repository

For some repositories, it may be desirable to think of them as "temporary storage". Thus it is desirable to keep the repository itself, but not the items contained within it. To clear out a repository, right-click on it and a context menu appears. From this pop-up menu, choose "Delete Contents". The items in the repository are then removed, while the repository itself remains available for future use.

Adding a Repository

A new repository can be added by choosing "New…" from the Repositories window.

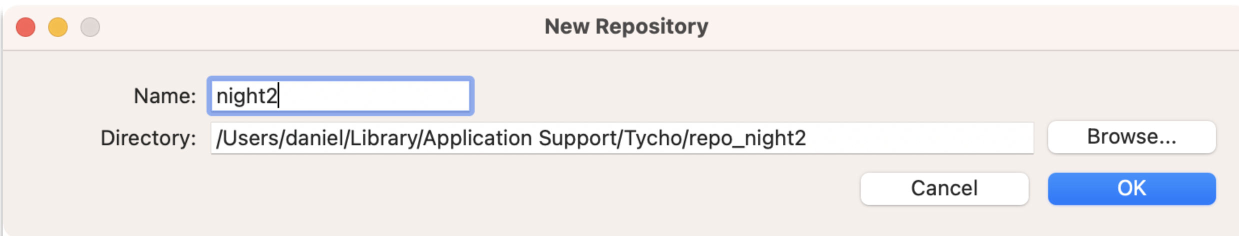


Figure 65 - Adding a Repository

Removing a Repository

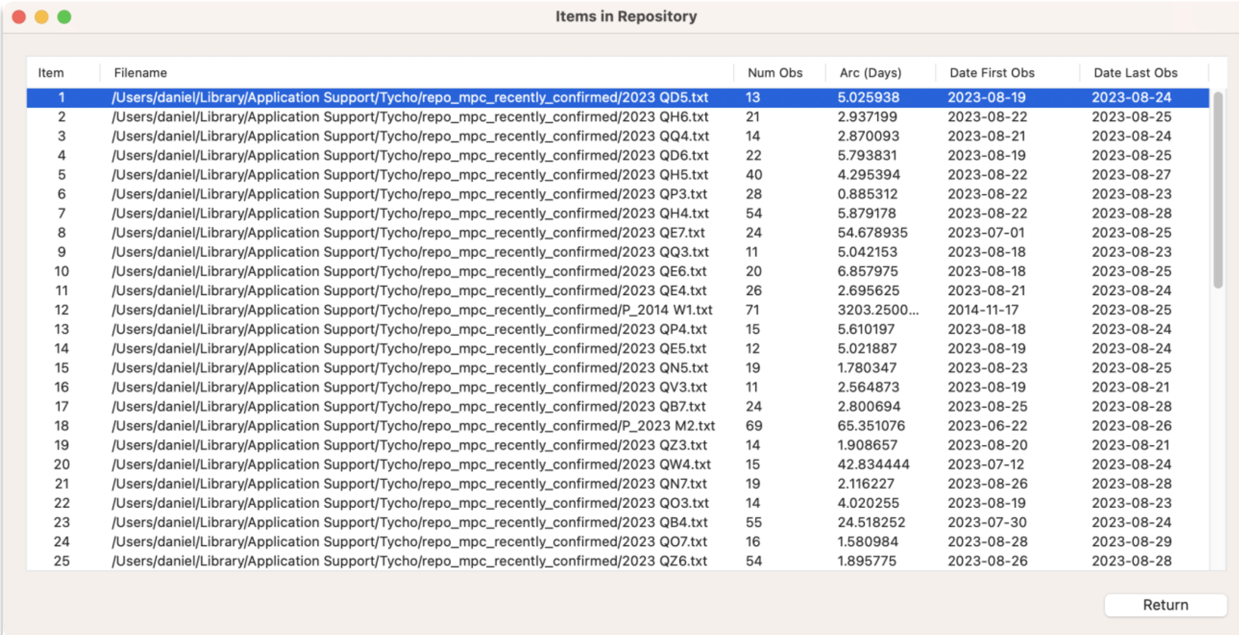
A repository can be removed from the list of repositories by selecting it from the list and clicking the "Remove Selected" button from the Repositories window. Alternatively, one could also right-click the repository and choose "Remove from List". Note that removing a repository simply removes it from the list – it does not delete the directory associated with the repository. To empty the contents of a repository, follow the steps in "Clearing a Repository". To delete the actual directory itself, choose "Open in Explorer" (or "Open in Finder" for macOS) and delete the directory.

Refresh Repository List

Most actions, such as adding or removing a repository, will automatically refresh the list of repositories as well as their associated object count. However, if one were to manipulate the objects of a repository externally to the Repository Manager (such as via Explorer or Finder), then the object count may be out of sync. Simply click the "Refresh" button and all repositories along with the associated object count will be refreshed.

Viewing Items in a Repository

There are two ways to view the contents of a repository. One is to choose "Open in Explorer" ("Open in Finder" for macOS) which will reveal the standard directory window. The other method is to click the "Open…" button while in the Repositories window, which will present a new window showing each item in the repository along with the number of observations, the arc length, and the date of first and last observation for each item. While in this window, you can right-click on an object and choose "View" or "View Orbit". The first option, "View", will copy the observations of the object to the text window ("Text Form – Observations"). The latter option, "View Orbit", computes an orbit from the current set of measurements.



Item	Filename	Num Obs	Arc (Days)	Date First Obs	Date Last Obs
1	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QD5.txt	13	5.025938	2023-08-19	2023-08-24
2	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QH6.txt	21	2.937199	2023-08-22	2023-08-25
3	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QQ4.txt	14	2.870093	2023-08-21	2023-08-24
4	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QD6.txt	22	5.793831	2023-08-19	2023-08-25
5	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QH5.txt	40	4.295394	2023-08-22	2023-08-27
6	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QP3.txt	28	0.885312	2023-08-22	2023-08-23
7	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QH4.txt	54	5.879178	2023-08-22	2023-08-28
8	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QE7.txt	24	54.678935	2023-07-01	2023-08-25
9	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QQ3.txt	11	5.042153	2023-08-18	2023-08-23
10	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QE6.txt	20	6.857975	2023-08-18	2023-08-25
11	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QE4.txt	26	2.695625	2023-08-21	2023-08-24
12	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/P_2014 W1.txt	71	3203.2500...	2014-11-17	2023-08-25
13	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QP4.txt	15	5.610197	2023-08-18	2023-08-24
14	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QE5.txt	12	5.021887	2023-08-19	2023-08-24
15	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QN5.txt	19	1.780347	2023-08-23	2023-08-25
16	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QV3.txt	11	2.564873	2023-08-19	2023-08-21
17	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QB7.txt	24	2.800694	2023-08-25	2023-08-28
18	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/P_2023 M2.txt	69	65.351076	2023-06-22	2023-08-26
19	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QZ3.txt	14	1.908657	2023-08-20	2023-08-21
20	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QW4.txt	15	42.834444	2023-07-12	2023-08-24
21	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QN7.txt	19	2.116227	2023-08-26	2023-08-28
22	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QO3.txt	14	4.020255	2023-08-19	2023-08-23
23	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QB4.txt	55	24.518252	2023-07-30	2023-08-24
24	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QO7.txt	16	1.580984	2023-08-28	2023-08-29
25	/Users/daniel/Library/Application Support/Tycho/repo_mpc_recently_confirmed/2023 QZ6.txt	54	1.895775	2023-08-26	2023-08-28

Figure 66 - Viewing Items in a Repository

Identify Linkages

Another new feature in v5.0 is the ability to identify linkages using a new "Object Linker" interface. This module works alongside the repository feature in that it allows one to simply specify a repository that contains the desired objects for link detection.

As an example, consider the scenario of trying to determine whether or not a newly detected object is already listed on the NEOCP or "Recently Confirmed" page. The first step is to update these repositories (discussed earlier) by going to ***Actions->Download NEOCP/Recently Confirmed*** via the "Repositories" window. Once these repositories have been populated, they can then be used in the Object Linker tool. The next step is to then choose these two repositories as the "A" search path, and then choose the repository containing the newly detected object as the "B" search path. Refer to Figure 67.

Once the link process has completed, the results are shown in the "Link Results" window. Up to 5 links are shown per each object, depending on the setting "Links per object". In this case, since there is only one object in the "B" search path, there are exactly five results shown for applied setting. The results are sorted by median residual. A median residual of one arcsecond or less (<1.00) is usually indicative of a possible link. As shown in Figure 68, the object "DJP1001" links up with NEOCP object "P21l1Bp". By right-clicking on the link, one can view each object as well the result of combining the observations of each object. It is also possible to save the combined result to its own separate repository.

Another scenario for the Object Linker tool is to determine what links, if any, exist between two nights of data from the same field. For example, you might have wondered if an object you detected on the first night could really be found on the second night. Normally, you would use ephemeris information and recover the object by setting the tracking parameters around that information. But in the case where you have perhaps dozens or more objects to confirm between the two nights, it can be easier to use the Object Linker tool.

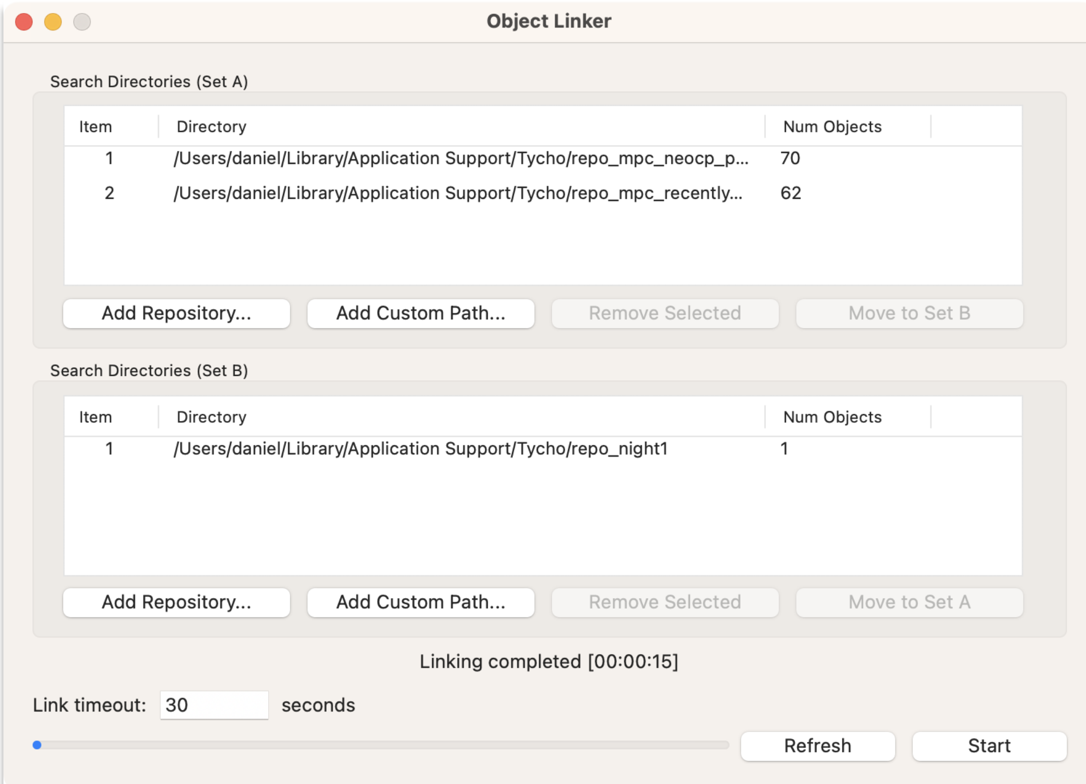


Figure 67 - Object Linker

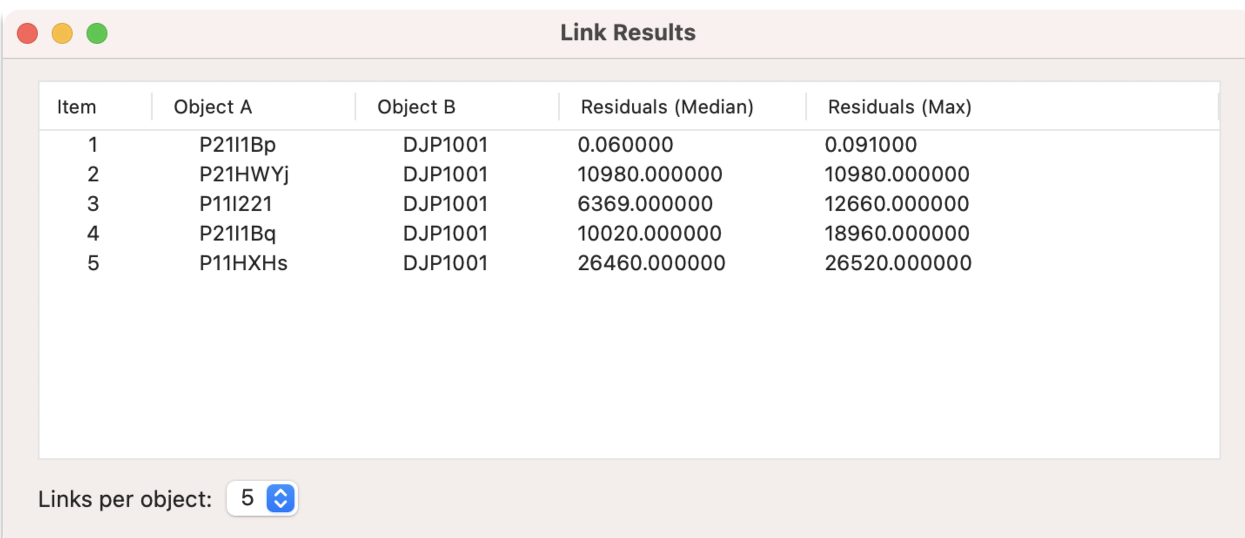


Figure 68 - Link Results

Continuing with this example, you could create the relevant observations for the first night and store them in their own repository (e.g., "Night1"). Then, do the same for the second night and store them in a repository, "Night2". Next, use the Object Linker tool (**Tools->Identify Linkages**) and specify "Night1" repository as Set A, and the "Night2" repository as Set B. Run the linker, and examine the results. For each link having an acceptable median residual (typically < 1.00), keep it in the list. Otherwise, if the link is not acceptable, delete it (right-click on the link(s) and choose "Delete"). Once you have eliminated all invalid links, go back to the "Observations -- All Targets" window (or go to ***File->Load Observations*** if it is not currently available) and merge both nights of observations. For example, if the Observations window is currently showing the observations from Night2, choose "**File->Load Observations (Append to Existing)**" and choose Night1 observations to be appended to the list. Once the Observations window is showing the combined observations of both nights, and you have the "Link Results" window showing only the valid links between those two nights, you can then proceed to choose ***Observations->Apply Confirmed Links*** from the Observations window and this will give each object that has an associated link a shared designation. Finally, if you want to eliminate the observations for which there is no confirmed link, you can choose ***Observations->Remove Unconfirmed***.

Image Statistics

Another new feature introduced in v5.0 is the ability to generate statistics for a given image, be it a single exposure or a stack of exposures. These statistics can be generated by choosing ***File->Generate Stats for Current Image…*** from the "Image Viewer". It will prompt for the max number of stars to use, with 2000 being the default. The stars are selected either at random or in a fixed manner from the loaded catalog. It is recommended to use the Gaia DR2 catalog to ensure reasonable depth of detection for images that reach past magnitude 18. Once you have specified the max number of stars, click "Start" and the tool will begin collecting statistics about the current image shown in the Image Viewer. Again, this could be a stacked image – for example, by choosing "**Selection->All**" from the Image Manager, you would create a stack of all images. Or it could be a single exposure. Either way, once the statistics have been generated, the results are shown in the Image Statistics window. Refer to Figure 69.

Note: Be sure that the image is *not* derived from tracked motion. In other words, it must either be a single image, or a stack of images with zero motion applied (the stars should not be streaked).

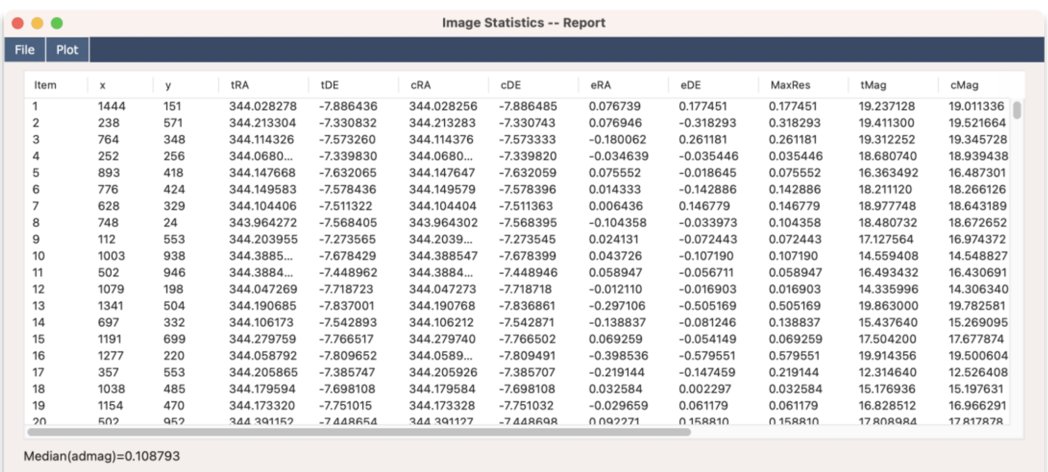


Image Statistics -- Report

File | Plot

Item	x	y	tRA	tDE	cRA	cDE	eRA	eDE	MaxRes	tMag	cMag
1	1444	151	344.028278	-7.886436	344.028256	-7.886485	0.076739	0.177451	0.177451	19.237128	19.011336
2	238	571	344.213304	-7.330832	344.213283	-7.330743	0.076946	-0.318293	0.318293	19.411300	19.521664
3	764	348	344.114326	-7.573260	344.114376	-7.573333	-0.180062	0.261181	0.261181	19.312252	19.345728
4	252	256	344.0680...	-7.339830	344.0680...	-7.339820	-0.034639	-0.035446	0.035446	18.680740	18.939438
5	893	418	344.147668	-7.632065	344.147647	-7.632059	0.075552	-0.018645	0.075552	16.363492	16.487301
6	776	424	344.149583	-7.578436	344.149579	-7.578396	0.014333	-0.142886	0.142886	18.211120	18.266126
7	628	329	344.104406	-7.511322	344.104404	-7.511363	0.006436	0.146779	0.146779	18.977748	18.643189
8	748	24	343.964272	-7.568405	343.964302	-7.568395	-0.104358	-0.033973	0.104358	18.480732	18.672652
9	112	553	344.203955	-7.273565	344.2039...	-7.273545	0.024131	-0.072443	0.072443	17.127564	16.974372
10	1003	938	344.3885...	-7.678429	344.388547	-7.678399	0.043726	-0.107190	0.107190	14.559408	14.548827
11	502	946	344.3884...	-7.448962	344.3884...	-7.448946	0.058947	-0.056711	0.058947	16.493432	16.430691
12	1079	198	344.047269	-7.718723	344.047273	-7.718718	-0.012110	-0.016903	0.016903	14.335996	14.306340
13	1341	504	344.190685	-7.837001	344.190768	-7.836861	-0.297106	-0.505169	0.505169	19.863000	19.782581
14	697	332	344.106173	-7.542893	344.106212	-7.542871	-0.138837	-0.081246	0.138837	15.437640	15.269095
15	1191	699	344.279759	-7.766517	344.279740	-7.766502	0.069259	-0.054149	0.069259	17.504200	17.677874
16	1277	220	344.058792	-7.809652	344.0589...	-7.809491	-0.398536	-0.579551	0.579551	19.914356	19.500604
17	357	553	344.205865	-7.385747	344.205926	-7.385707	-0.219144	-0.147459	0.219144	12.314640	12.526408
18	1038	485	344.179594	-7.698108	344.179584	-7.698108	0.032584	0.002297	0.032584	15.176936	15.197631
19	1154	470	344.173320	-7.751015	344.173328	-7.751032	-0.029659	0.061179	0.061179	16.828512	16.966291
20	502	952	344.391152	-7.448654	344.391127	-7.448698	0.092271	0.158810	0.158810	17.808984	17.817878

Median(admag)=0.108793

Figure 69 - Image Statistics

A number of data points are collected for each star in the image, including RA and Declination residuals, true magnitude versus calculated magnitude, flux, FWHM, and SNR. Clicking on a row in the list will automatically navigate to the associated star in the Image Viewer. It is also possible to generate plots for some of these fields from the "Plot" menu. As an example, the "MaxRes vs Mag" plot will show a graph indicating the relationship between magnitude and max residual, with the fainter magnitude stars typically having a higher maximum residual (as expected). Using this graph, it is also possible to get an idea of the limiting magnitude of the image, with the text in the lower-right corner indicating the % of data points satisfying the threshold at a given magnitude. As an example, a single exposure from the dataset "ds2" is shown in Figure 70.

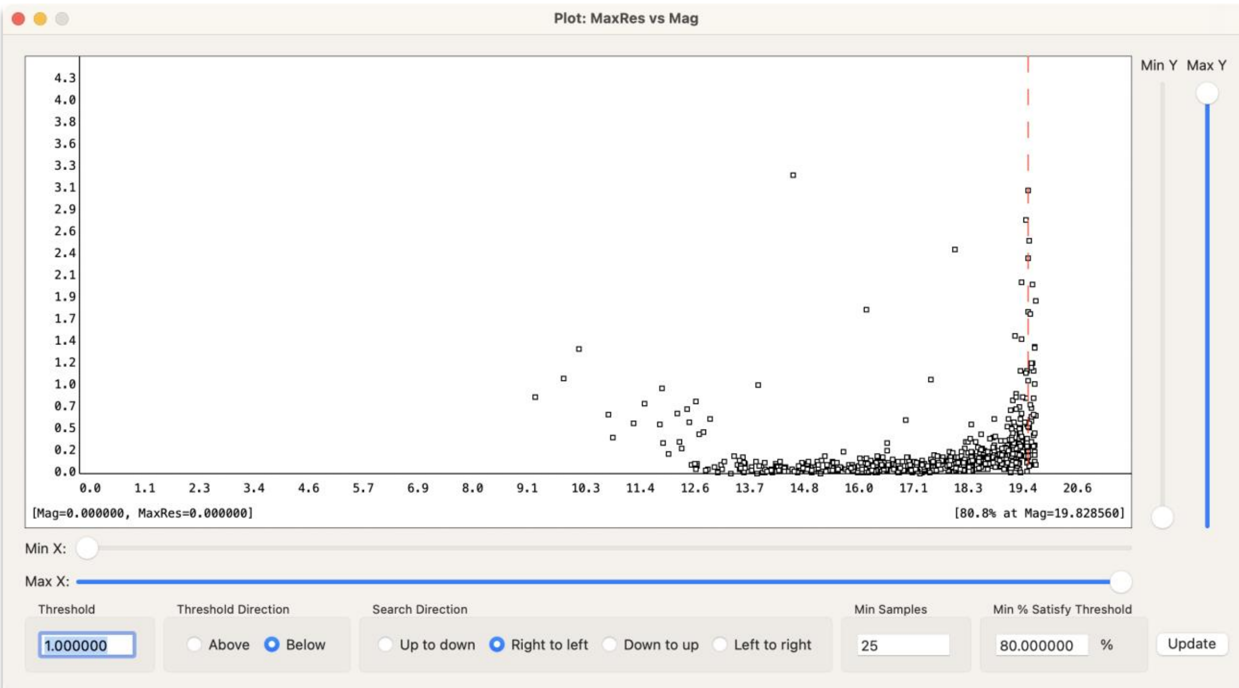


Figure 70 - Plot of MaxRes vs Magnitude

It is also possible to export the data to a CSV file by choosing ***File->Save to CSV*** from the "Image Statistics" window. This enables additional work to be done on the data through other programs such as Excel.

One particularly useful feature of the "Image Statistics" tool is the ability to easily determine if the image has a good plate solution attached to it, or has some other issue that would result in higher than acceptable residuals. This can be accessed by going to ***Plot->Median Residual*** from the "Image Statistics" window. From here, you have the ability to choose the number of divisions that comprise the grid. In the dropdown labeled "Num divisions:", choose "4" as the setting and you can see the image divided up into a 4x4 grid, with each cell indicating the residual for that region of the image. An example of a dataset that has excellent residuals across the entire image is shown in Figure 71, whereas an example of a dataset with poor residuals is shown in Figure 72.

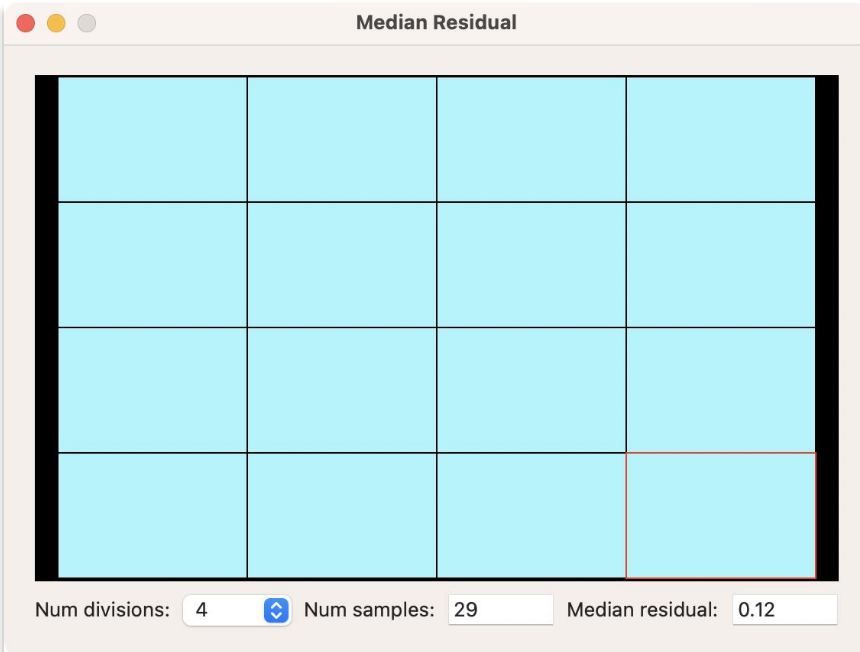


Figure 71 - Example of Dataset with Excellent Residuals

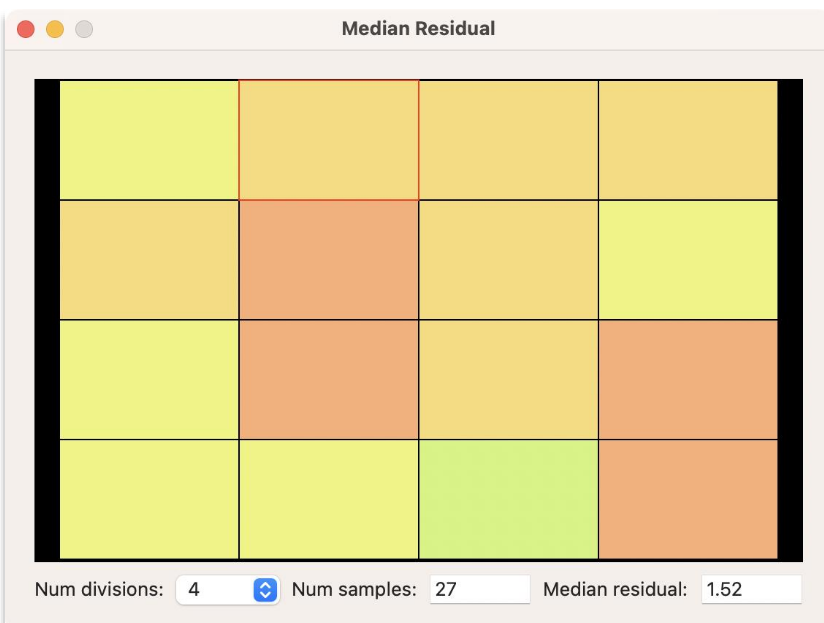


Figure 72 - Example of Dataset with Poor Residuals

An example of a dataset having poor residuals is shown In Figure 72. In summary, you can use this tool to help validate both the quality of the images as well as the quality of the plate solution. If either one has subpar quality, then that should show up in the grid display. Note that there is also another image evaluation module available via **Action->Evaluate Images** from the main menu.

Constructing a Lightcurve

Tycho (Professional) also offers the ability to construct lightcurves and determine rotation periods.

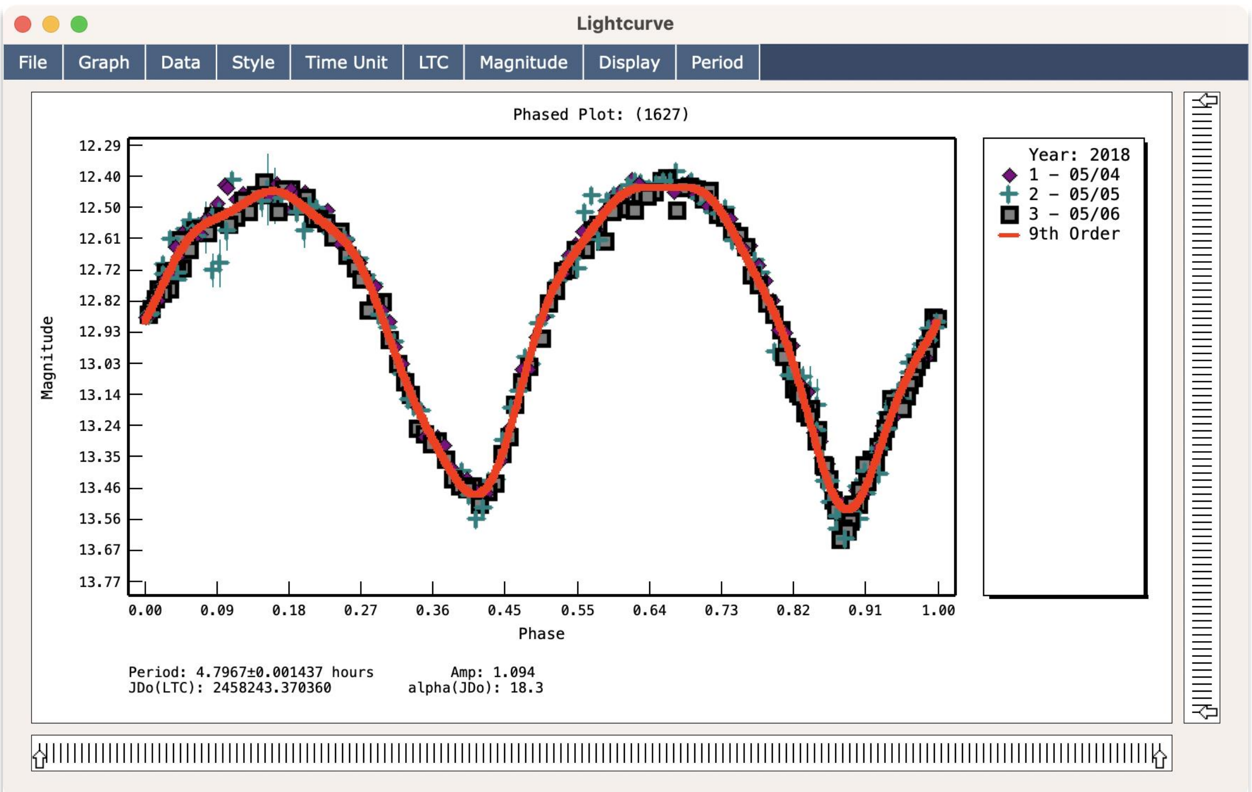


Figure 73 - Lightcurve of asteroid (1627) Ivar

Step 1: Update the "Active Observatory"

The datasets used by this example were captured by Platanus Observatory, which has MPC code "K80". Navigate to ***Settings->Observatory*** from the main menu. Then choose ***Action->Add Observatory*** from the menu of the "Observatory Configuration" window. It will prompt for the observatory details. Specify "K80" (without quotes) for the "Label" field. Choose "Already have an MPC code" for the observatory status. Then specify "K80" (again without quotes) for the MPC code. Click the button "Apply location from MPC code" to automatically populate the location information. The "Telescope" field is not important for this example, but you can use the same settings as shown in Figure 74. Then click the "Next…" button to continue to the next page.

On the second page, make sure your settings match those shown in Figure 75. Then click "Finished" to add the new observatory.

Find the new observatory in the list. Then right-click on it and choose "Make Active" from the popup menu that appears.

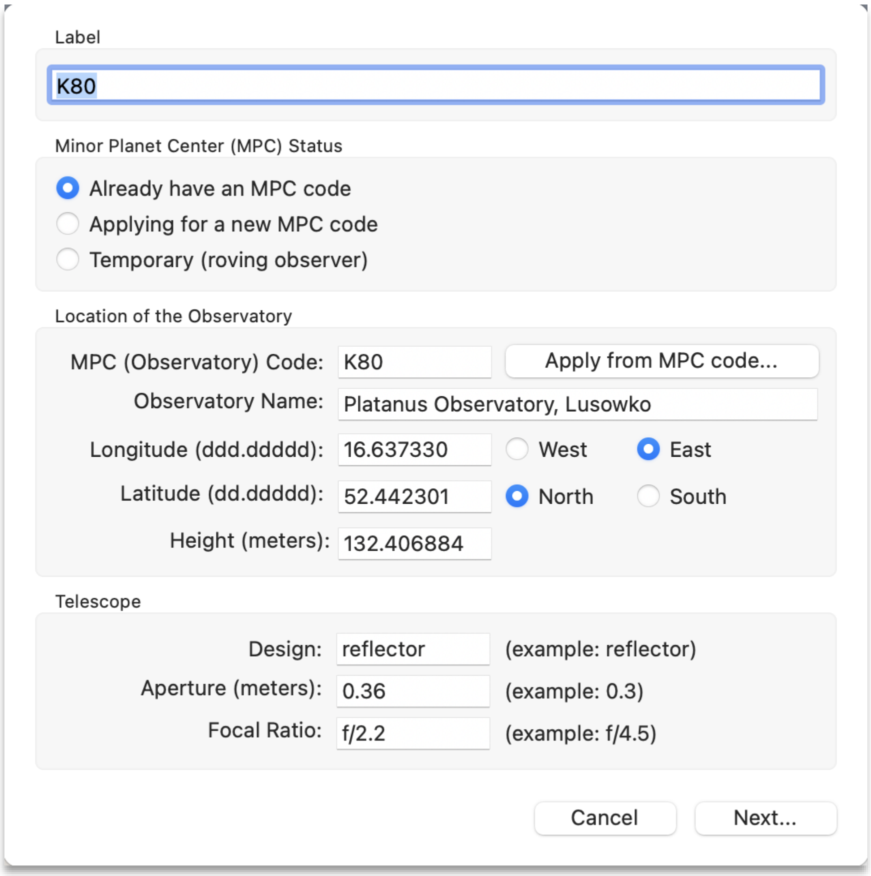


Figure 74 - Configuring Observatory 1/2

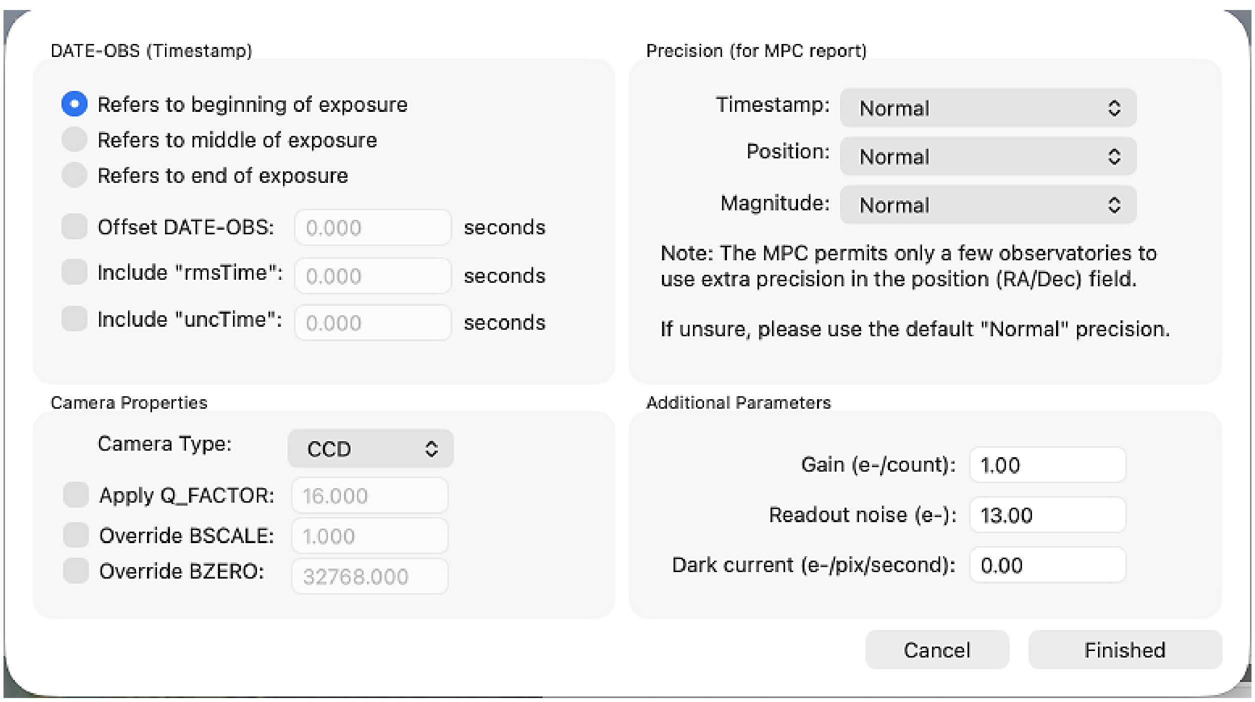
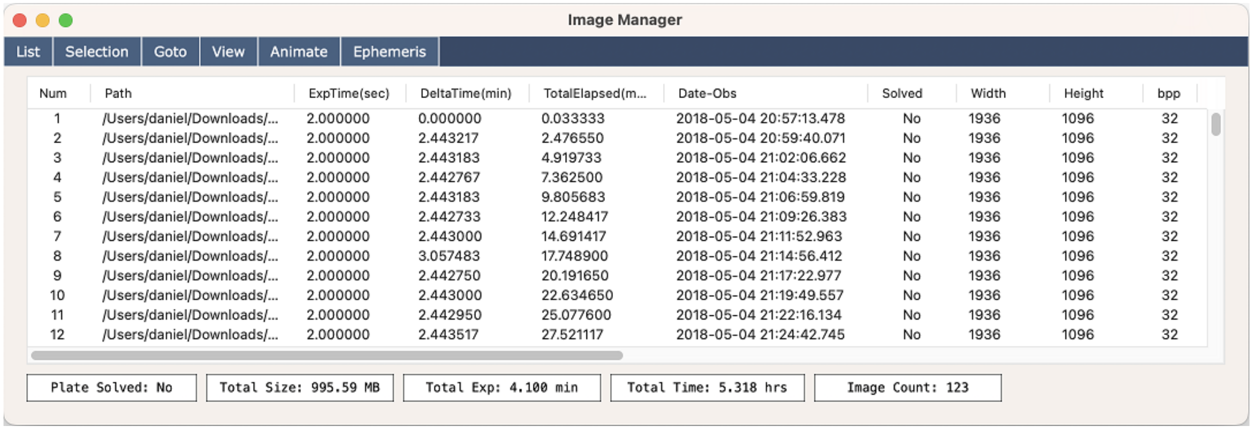


Figure 75 - Configuring Observatory 2/2

Step 2: Load the "Ivar Night 1" Dataset

Launch Tycho and navigate to ***List->Add Images*** from the "Image Manager". The images for this example are located in the "n1" subdirectory of the dataset labeled "Ivar Lightcurve". There should be a total of 123 images in this dataset, which is the first of three nights of data collected for this object.



Num	Path	ExpTime(sec)	DeltaTime(min)	TotalElapsed(m...	Date-Obs	Solved	Width	Height	bpp
1	/Users/daniel/Downloads/...	2.000000	0.000000	0.033333	2018-05-04 20:57:13.478	No	1936	1096	32
2	/Users/daniel/Downloads/...	2.000000	2.443217	2.476550	2018-05-04 20:59:40.071	No	1936	1096	32
3	/Users/daniel/Downloads/...	2.000000	2.443183	4.919733	2018-05-04 21:02:06.662	No	1936	1096	32
4	/Users/daniel/Downloads/...	2.000000	2.442767	7.362500	2018-05-04 21:04:33.228	No	1936	1096	32
5	/Users/daniel/Downloads/...	2.000000	2.443183	9.805683	2018-05-04 21:06:59.819	No	1936	1096	32
6	/Users/daniel/Downloads/...	2.000000	2.442733	12.248417	2018-05-04 21:09:26.383	No	1936	1096	32
7	/Users/daniel/Downloads/...	2.000000	2.443000	14.691417	2018-05-04 21:11:52.963	No	1936	1096	32
8	/Users/daniel/Downloads/...	2.000000	3.057483	17.748900	2018-05-04 21:14:56.412	No	1936	1096	32
9	/Users/daniel/Downloads/...	2.000000	2.442750	20.191650	2018-05-04 21:17:22.977	No	1936	1096	32
10	/Users/daniel/Downloads/...	2.000000	2.443000	22.634650	2018-05-04 21:19:49.557	No	1936	1096	32
11	/Users/daniel/Downloads/...	2.000000	2.442950	25.077600	2018-05-04 21:22:16.134	No	1936	1096	32
12	/Users/daniel/Downloads/...	2.000000	2.443517	27.521117	2018-05-04 21:24:42.745	No	1936	1096	32

Figure 76 - The "Ivar_n1" Dataset with 123 Images

Step 3: Perform "Express Mode"

Navigate to ***Action->Express Mode*** and click the "Settings…" button located to the left of "Plate Solve". If you have not yet configured the plate solver, proceed to do so now. Refer to the section labeled "Configure the Plate Solver" for details. An example configuration is shown in Figure 78. Click "Save Settings" to continue.

Navigate to ***Action->Express Mode*** and click the "Settings…" button located to the left of "Align". An example configuration is shown in Figure 77. Then click "Save Settings" to continue.

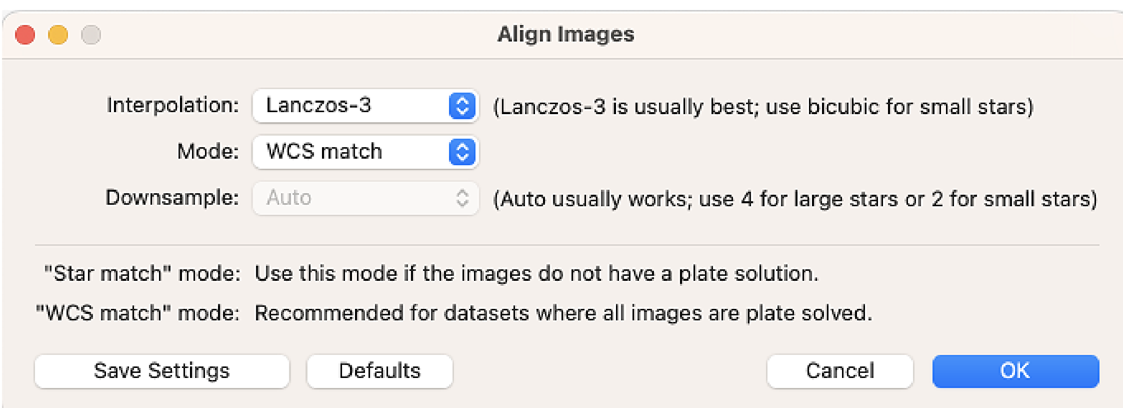


Figure 77 – Alignment Settings

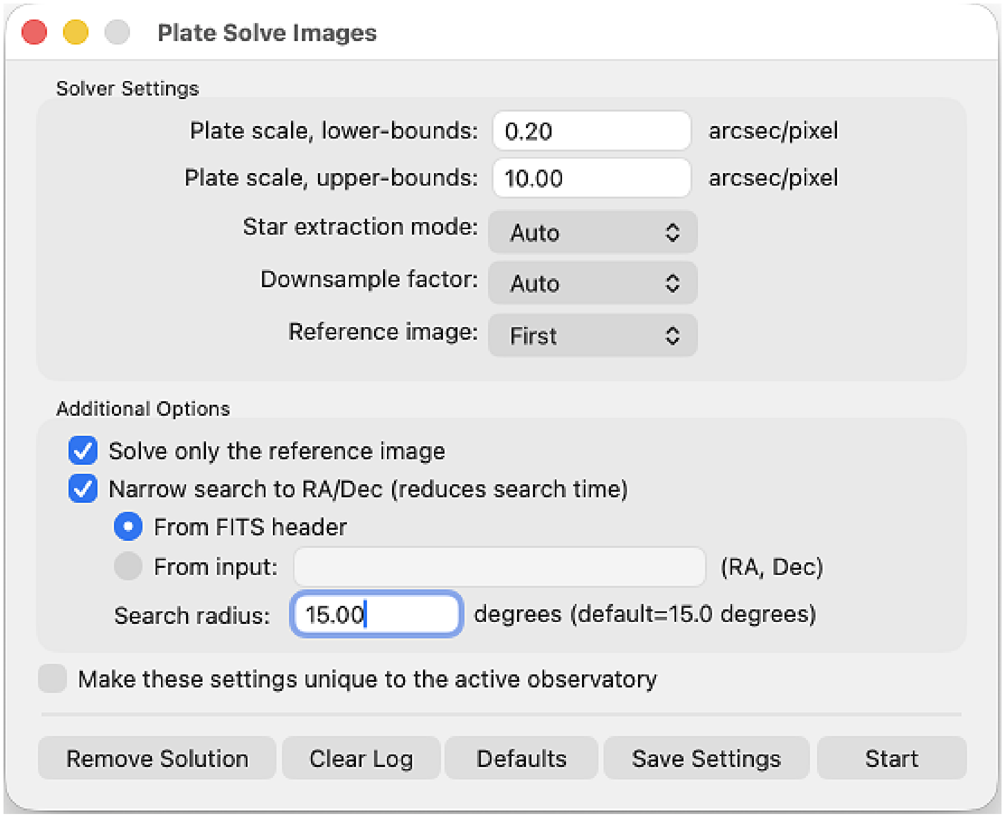


Figure 78 – Plate Solve Configuration

Then, verify that your “Express Mode” settings match those shown in Figure 79.

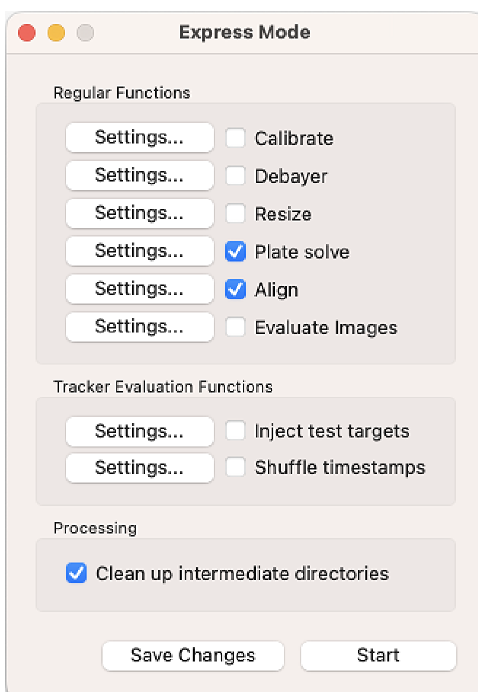


Figure 79 - Express Mode Settings

At this point, you should be ready to process the images for the first night (n1). Click the "Start" button on the "Express Mode" window to proceed. After a minute, the images should be plate solved and aligned.

From the "Image Manager", navigate to ***List->Add Images*** and load the images from the output directory, "n1_a". You should once again have approximately 120 images loaded, except this time they are now processed and ready for measurements to be created. Before moving onto the next step, navigate to ***Action->View Images*** from the main menu.

Step 4: Adjust the Apertures

The default photometry apertures are 4.0 pixels for the inner annulus, 2.0 pixels for the dead-zone (region between inner and outer annulus) and 9.0 pixels for the outer annulus. To adjust the apertures, choose ***Photometry->Modify Aperture Settings*** from the "Image Viewer" menu. Here a new window will appear, where you can adjust the aperture settings. You will also note that the crosshairs have changed to indicate the current aperture. An optimal inner annulus will include most of the light from the target and a minor amount from the background sky. When satisfied with the setting, click "Close".

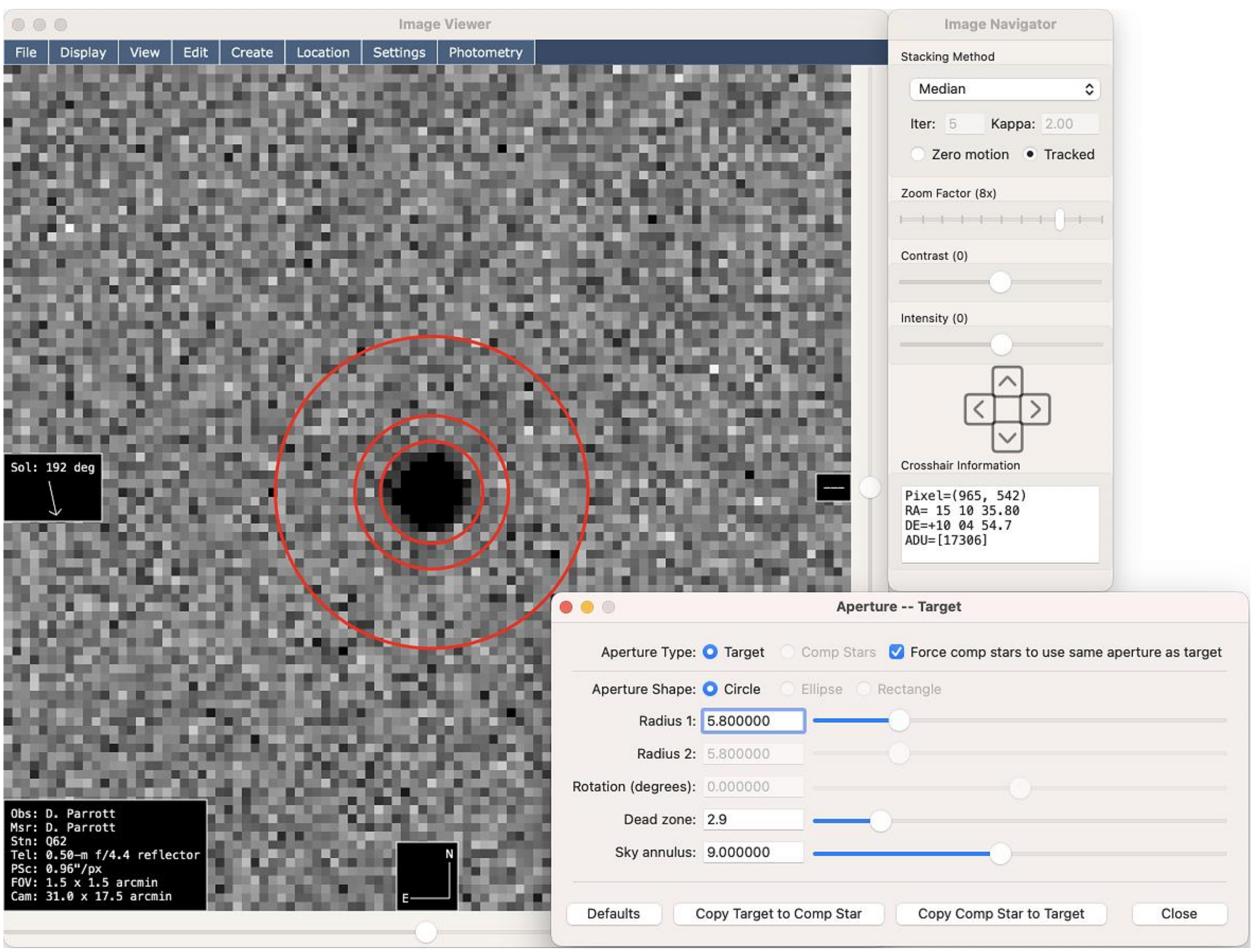


Figure 80 - Adjusting Aperture Settings

As shown in Figure 80, the radius has been adjusted to be 5.8 pixels for the inner aperture and 2.9 pixels for the Dead zone. It is also a good idea to click the button "Copy Target to Comp Star", so that the comparison stars use the same aperture as that of the target object. However, this is not necessary if the box labeled "Force comp stars to use same aperture as target" is checked. Again, the optimal aperture should be determined by the target. If you are working on a known asteroid, you can find it in the image by selecting ***File->Load Known Objects*** from the "Image Viewer". For variable stars, you can choose ***Photometry->Variable Stars*** from the "Image Viewer".

Step 5: Specify Comparison Stars

New in v8.0 is the ability to manually specify which stars are used as comparison stars (comp stars) for the photometry measurements. If you wish to have Tycho automatically choose comp stars, then you can skip this step. Otherwise, proceed to navigate to ***Photometry->Find Comp Stars*** from the menu of the "Image Viewer". You should see a new window appear that resembles the one shown in Figure 81.

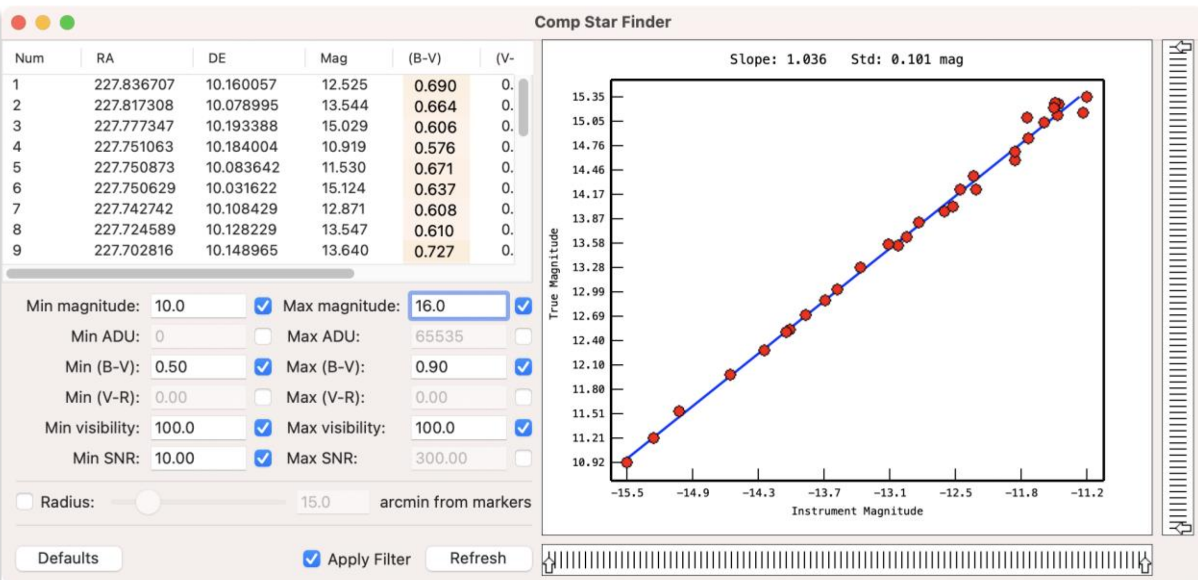


Figure 81 - Identify Optimal Comparison Stars

On the right half of the window there is a graph showing a trend line. This indicates the relationship between instrument magnitude and true magnitude for each star that meets the filter specified in the left half of the window. Ideally, the slope of this line would be 1.00 and the standard deviation (Std) would be 0.00, indicating a perfect fit of the data.

One common reason for a larger deviation is if you have the "Max magnitude" filter set too high. These exposures are only 2 seconds, so the default max of 20.0 includes a lot of very faint stars. To remedy this, specify a lower value for "Max Magnitude" (such as 16.0) and then click the "Refresh" button. The graph will update accordingly.

You can also filter stars by solar temperature, specifying a desired lower- and upper-bound for (B-V). The Sun has a (B-V) value of around 0.6, and it is usually common to set the min and max to 0.5 and 0.9 respectively. Alternatively, if you are working in (V-R), you could instead apply a filter for that range of temperatures. The other filters work in a similar fashion with a min and max setting.

The radius filter is a new option introduced in v8.1. It allows one to filter the comparison stars by proximity to the marker position(s). As an example, this might be useful if you wanted to select comparison stars that reside near the object. To apply the filter, first create a marker by going to ***Create->Marker 1*** from the “Image Viewer” menu, or right-clicking inside the image and choosing *Create Marker 1* from the popup menu. The marker will be created at the current location of the crosshairs. Next, click on the “Radius” checkbox to enable it, and then adjust the slider to set the desired radius from the marker. If you create both marker 1 and marker 2, then the center of the circle will be defined as the midpoint of the two markers. As an example, you might define marker 1 as being the first position of the object, and marker 2 as the last position of the object as it moves across the field. As with the other options, click the “Refresh” button to update the filter.

Each star that “survives” the resulting filter is then shown on the trend line as a red circle. You can click on a red circle and Tycho will then center the star in the “Image Viewer”. If you want to use the given star as a comparison star for photometry, you can then right-click in the graph and choose “Add to Active Comp Stars” from the popup menu that appears. Alternatively, you could zoom in and double-click on the star in the “Image Viewer”, right-click, and choose “Add to Active Comp Stars” from the popup menu that appears. Both approaches accomplish the same result.

If you want to see more information about a star, double-click on it in the “Image Viewer” to center it, then right-click and choose “View Star Information” from the popup menu that appears. This will present a new window as shown in Figure 82.

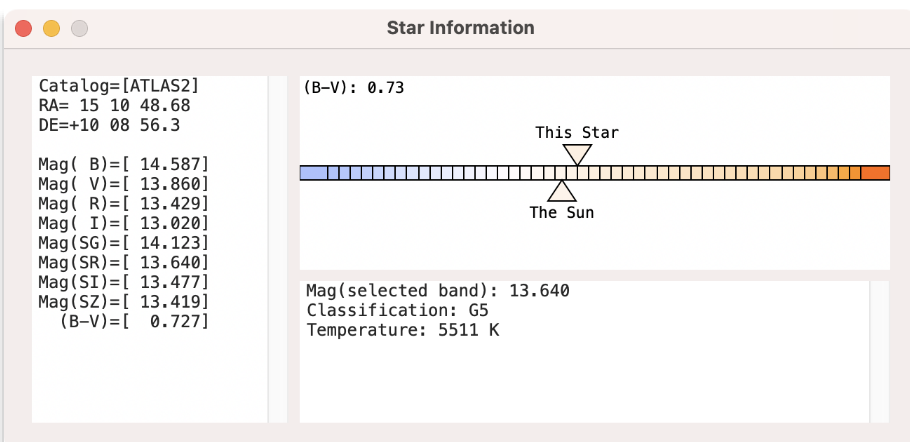


Figure 82 - View Star Information

Once you have an initial set of active comparison stars, you can navigate to ***Graph->Generate Data*** from the menu of the “Active Comparison Stars” window. This will measure each comp star across all of the images and generate graph information for each one as shown in Figure 83.

Be sure that “**Graph->Computed Mag vs Time**” is the current graph setting. If not, navigate to ***Graph->Computed Mag vs Time*** to set the option. For each comparison star, the graph shown on the right-half of the window should present a mostly flat, horizontal line of data points. This indicates that the star has a consistent measured magnitude throughout all the images in the dataset. If you see variability in the measured magnitude, then it is likely a good idea to discard it from the list. This can be done by right-clicking on the star in the list, and choosing “Remove” from the popup menu that appears.

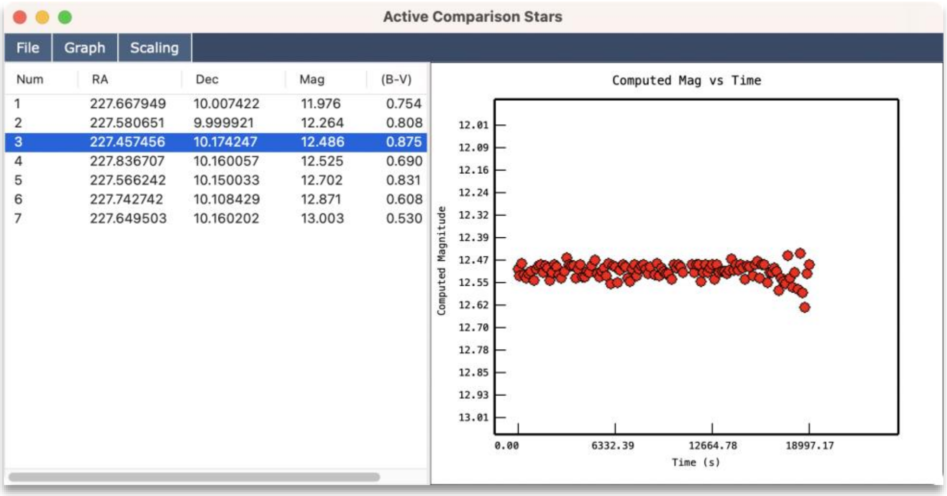


Figure 83 - Active Comparison Stars

It is usually common to choose around 5 to 7 comparison stars. Once you are satisfied with the list of active comparison stars, then you are ready to make photometry measurements. If you decide that you want to revert back to the "automatic" comp star selection, simply select all of the stars in the list, right-click, and choose "Remove". This will result in an empty list, and Tycho will assume that it therefore needs to identify its own set of comparison stars.

Step 6: Generate Photometry Measurements

The most straightforward approach to generating the measurements is to define two markers indicating the motion of the object. For this example, go to the "Image Manager" and click on the first image in the list, so that the first image is shown in the "Image Viewer". Then, locate the asteroid in the view and double-click on it to center it. If you are not sure where the asteroid is located, you could animate the frames, or you could attach ephemeris information and create a stack from ephemeris (refer to Example #4 earlier in this document). A third option is to invoke ***File->Load Known Objects*** from the "Image Viewer" menu. Be sure to click back on the first image if it is not selected. Then, once the object is centered (by double-clicking on it), right-click and choose "Create Marker 1" from the popup menu that appears. Next, choose the final image in the list (using the "Image Manager"), then locate the object in this last image. Once again, double-click the object to center it, and then right-click and choose "Create Marker 2" from the popup menu. At this point, you should now have two markers defined, indicating the motion of the object. For objects that have no motion (such as stars), you would simply define both markers at the same location.

Now that the markers have been defined, right-click in the "Image Viewer" and choose "Create Photometry – From Markers". Depending on the number of images, this can take a moment to generate the measurements. Once finished, the set of measurements will be added to a new window, "Photometry Sets". It is also a good idea at this point to save your work, so proceed to ***File->Save to Repository*** from the "Photometry Sets" menu. You will be prompted to specify a name for the entry. In this example I chose "ivar_test" (without the quotes) for the name. Then click the "Save" button to save the data.

From the “Photometry Sets” menu, navigate to ***Graph->Plot all Sets*** and you will see a new window appear with the raw plot of the data you have taken.

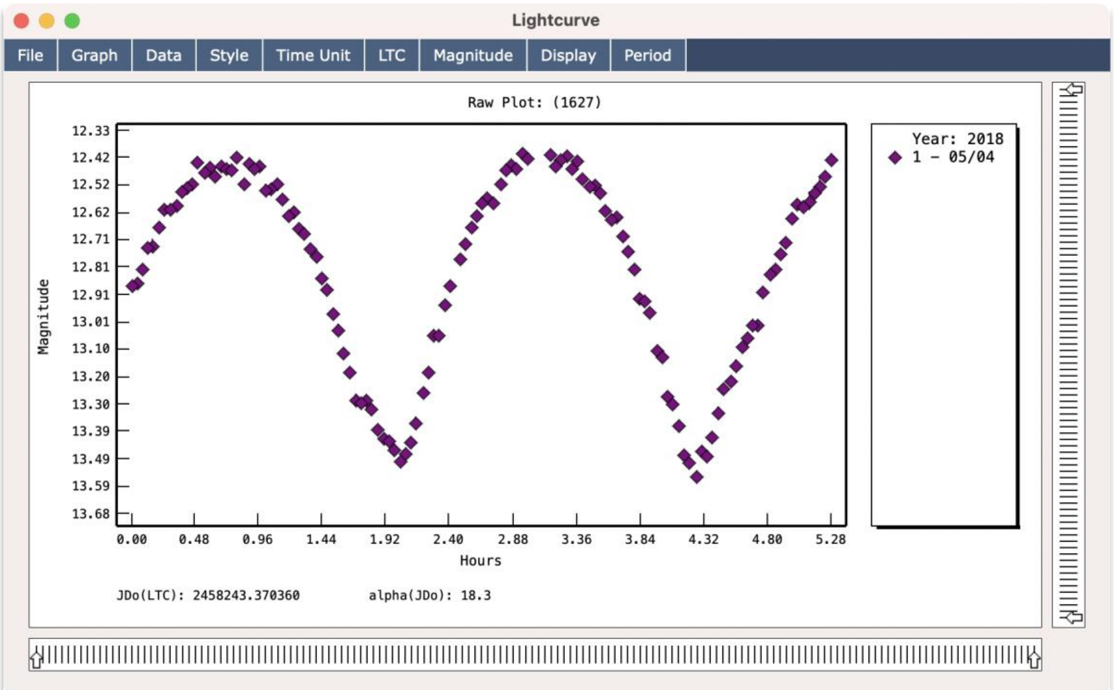


Figure 84 - First Night of Data for (1627) Ivar

In Figure 84, you can see the raw plot of the first night of data for Ivar. If desired, navigate to ***Data->Settings*** from the menu of the “Lightcurve” window and validate that the settings are as you want them to be (such as Exclude Outliers). If you ever need to delete data points, you can click and drag a rectangle selection around them, then right-click and choose “Delete” from the popup menu that appears. Or, you can right-click on them in the “Photometry Measurements” list and choose “Delete” from that popup menu.

If you made any changes to the data, you can save the photometry set again by going to ***File->Save to Repository*** from the menu of the “Photometry Sets” window. You can overwrite the previous entry or create a separate entry.

Step 7: Perform an Initial Period Search

If you were to conduct a period search with just this first night of data, you might get lucky and find the correct period. If you want to try, navigate to ***Period->Find Period*** from the menu of the “Lightcurve” window. A new window, “Period Search” appears. Make sure your settings match those in Figure 85 and then click the “Find Period” button to proceed.

You will note that from just this first night of data, the first candidate period is 2.36 hours, and the phased plot does not quite resemble a “bi-modal” curve (two valleys and two peaks). If you click on the second candidate period, 4.74 hours, then the phased plot resembles a bi-modal period, which suggests we are closer to finding the correct period. Using additional data (from nights 2 and 3) you will see that the correct period of 4.795 hours will be found with no issue.

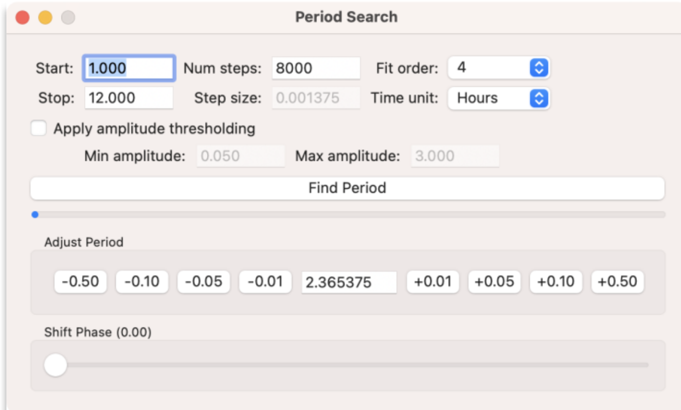


Figure 85 - Period Search Settings

As mentioned earlier, you can click on the different "candidate periods" to automatically refresh the phased plot display. These candidate periods are shown in the "Period List" window as indicated in Figure 86.

Period List

Num	Period (hours)	Amp	RMSE (x0.01 mag)	Factor (X)	Factor (Y)
1	2.365375	1.033	3.763940	1.000000	1.000000
2	4.745500	1.017	3.933370	2.006236	1.045014
3	5.829000	1.036	7.375705	2.464303	1.959570
4	9.901750	7.271	7.633093	4.186123	2.027953
5	5.325750	1.034	7.705006	2.251546	2.047059
6	7.234250	1.057	7.893401	3.058395	2.097111
7	3.271500	0.760	28.286147	1.383079	7.515036
8	1.167750	0.393	33.022384	0.493685	8.773355
9	1.327250	0.367	33.122764	0.561116	8.800024
10	1.275000	0.339	33.181066	0.539027	8.815514
11	1.613250	0.123	34.484849	0.682027	9.161901

Figure 86 - Period List

At this point you will want to include additional data for the period search. You will also want to use lighttime correction (LTC) and (H-G) correction. These steps are explained below.

Step 8: Include Nights 2 and 3

The previous steps showed how to process the data for the first night. Now we want to do similar and include nights 2 and 3 as additional photometry sets for this repository. The data for these nights are located in "ivar_n2" and "ivar_n3". Go ahead and create observations of the asteroid for night 2. If the resulting photometry set window contains only night 2, you can append night 1 by going to ***File->Load From Repository*** and choosing the repository you created earlier.

Proceed to save your progress by going to ***File->Save to Repository*** from the "Photometry Sets" menu. As before, you can overwrite the previous entry.

Now, as you did for night 2, do the same for night 3, so that you have three photometry sets.

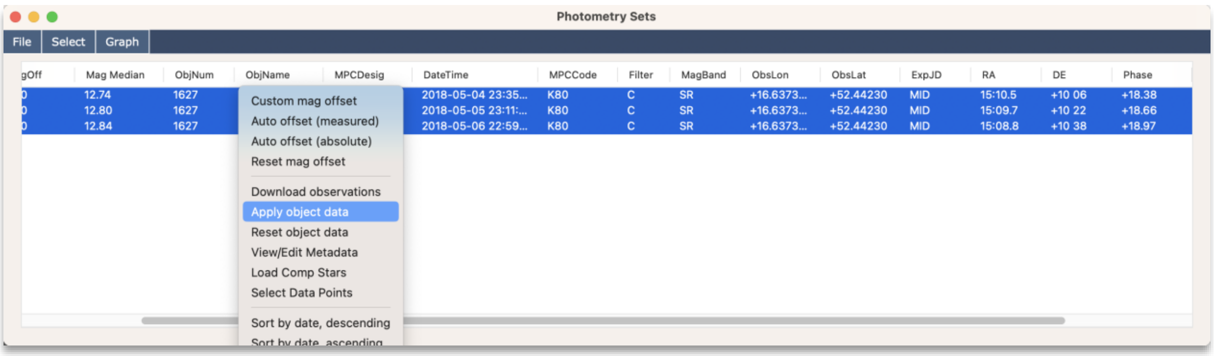


Figure 87 - Three Photometry Sets

Once you have the three photometry sets shown in the "Photometry Sets" window, you will want to apply object data to them. What this does is compute the Phase, Altitude, PABL, PABB, and Absolute Magnitude for every single observation. To do this, select the three sets, then right-click and choose "Apply object data" from the popup menu that appears. If you are not working with a minor planet, then you can skip this step.

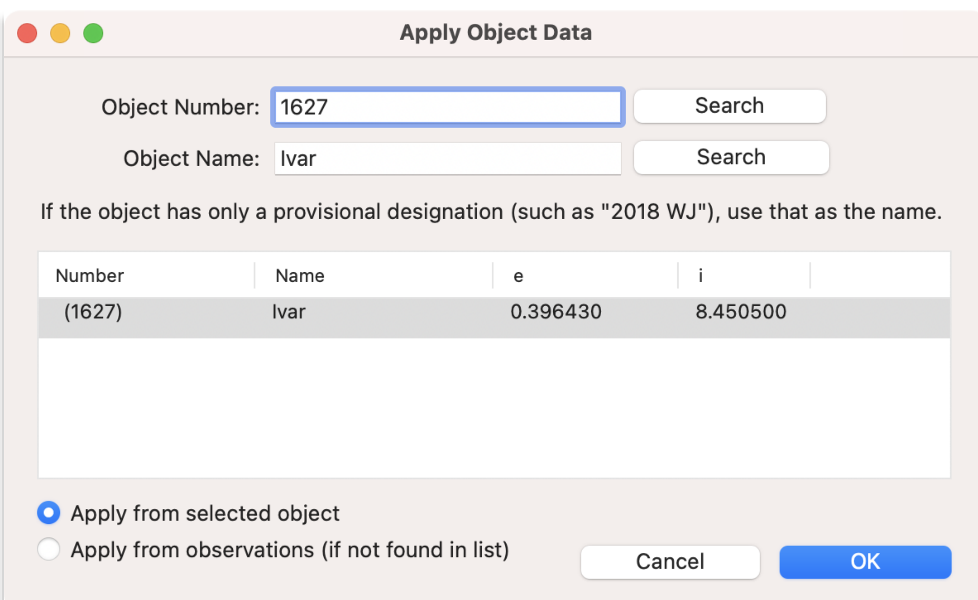


Figure 88 - Apply Object Data

A new window will appear, "Apply Object Data". Here you can specify the number and name of the object. If the orbital elements of the object are not yet established (such as a newly discovered object) then you can choose the option "Apply from observations" and click "OK". In this example, the object has number 1627, so you can specify that in the "Object Number" field and then click "Search". Then, click on the returned result and choose "Apply from selected object", and then click "OK" to proceed.

In the case where you might instead choose "Apply from observations", you will be prompted to instead choose ***Ephemeris->Attach to Photometry*** from the menu of an orbit window. If you do not already have an orbit loaded, go to ***Tools->Download Observations*** from the main menu, and specify the object. Then click "OK" to download the observations; a new window with the observations will appear. Then click "Compute Orbit". After a moment, a new window will appear with the computed orbit derived from the measurements. From this orbit window you can choose ***Ephemeris->Attach to Photometry*** and the currently selected photometry sets will be updated with the relevant object information.

At this point, your "Photometry Sets" window should now be similar to that shown in Figure 87. Once again, save your progress by going to ***File->Save to Repository***. As before, you can save over the previous entry.

Note: If you take several nights of data and find that a given night requires some sort of "offset" adjustment to match up with another night, you can either apply an automatic offset or a custom offset. For automatic offset, select the three photometry sets, right-click and choose "Auto offset (absolute)" from the popup menu that appears. For a custom offset, select the set of interest, right-click and choose "Custom mag offset". For these three nights, no offset is required.

Now, from the "Photometry Sets" window, navigate to ***Graph->Plot all Sets***. You will see a raw plot as shown in Figure 89. Because object data has been attached to these sets, both lighttime correction (LTC) and (H-G) correction are applied by default. You can check this by navigating to either the "LTC" menu (for lighttime correction) or the "Magnitude" menu (for "H-G" correction). Using (H-G) correction is typically important when performing period analysis, as it normalizes the measurements across phase angle and distance. Refer to https://www.britastro.org/asteroids/dymock4.pdf for more details on (H-G) correction.

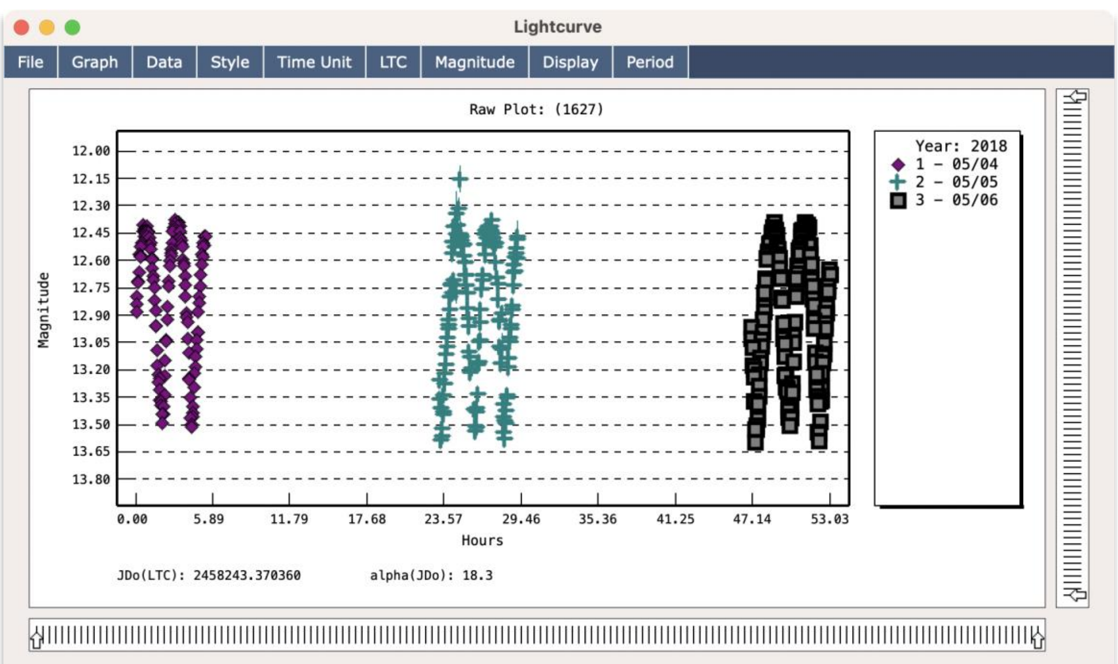


Figure 89 - Raw Plot of 3 Nights

Step 9: Final Period Search

Navigate to ***Period->Find Period*** from the “Lightcurve” menu. Use the settings as shown in Figure 90.

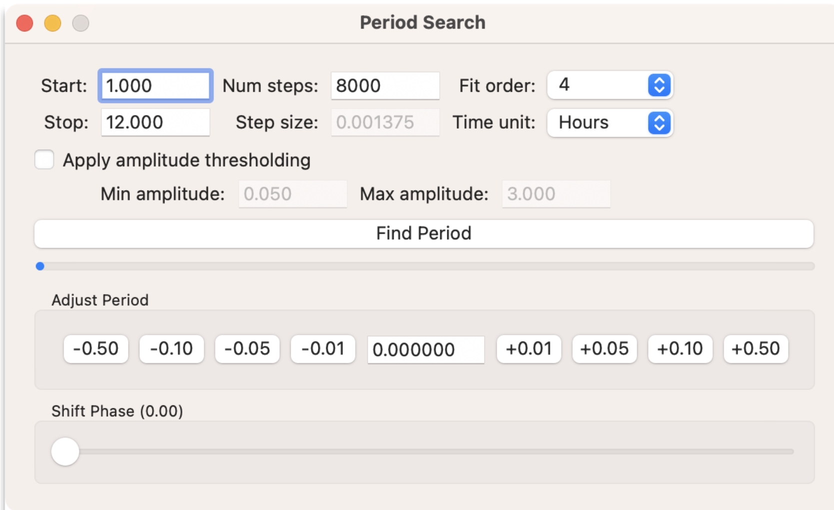


Figure 90 - Period Search Settings

Click the “Find Period” button and you will notice that this time, with all three nights of data available, Tycho is able to find the correct period (4.795 hours) and also lists it as the first candidate. It also displays a period spectrum, or “periodogram”, which is a graph showing the other possible periods throughout the search interval of 1-12 hours. If you want, you can also re-run the period search using a different Fit order. Figure 92 shows an example of a 9th order fit to the data. Usually, 4th order is a good starting point, and you want to avoid overfitting the data.

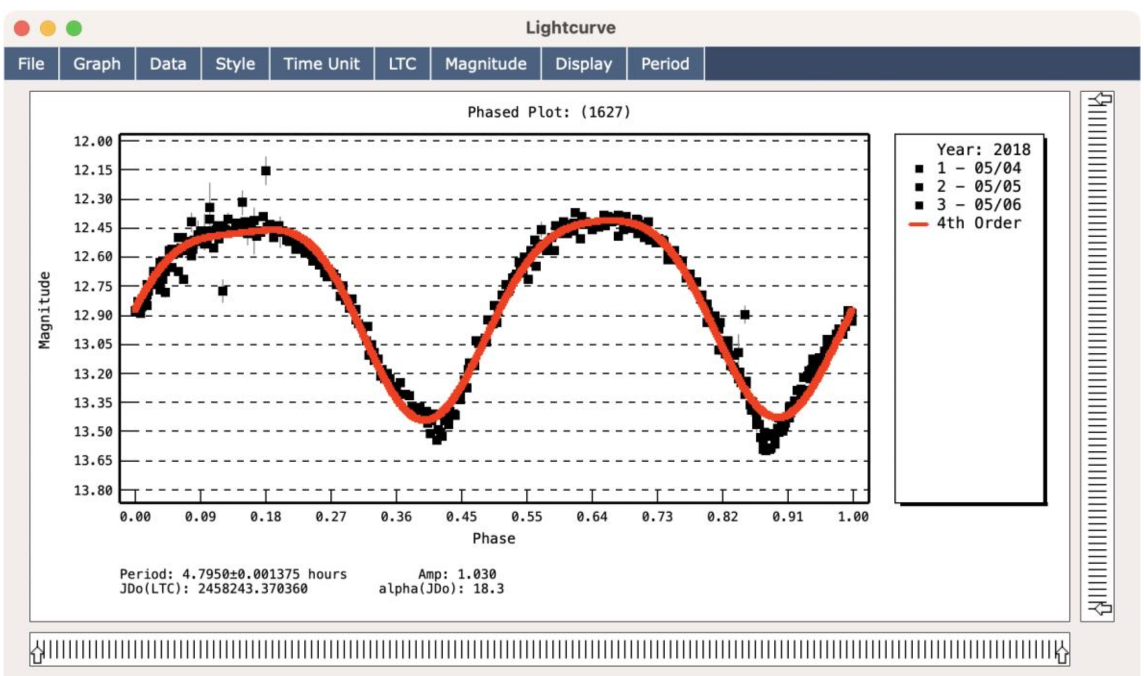


Figure 91 - Phased Plot of (1627) Using 3 Nights of Data

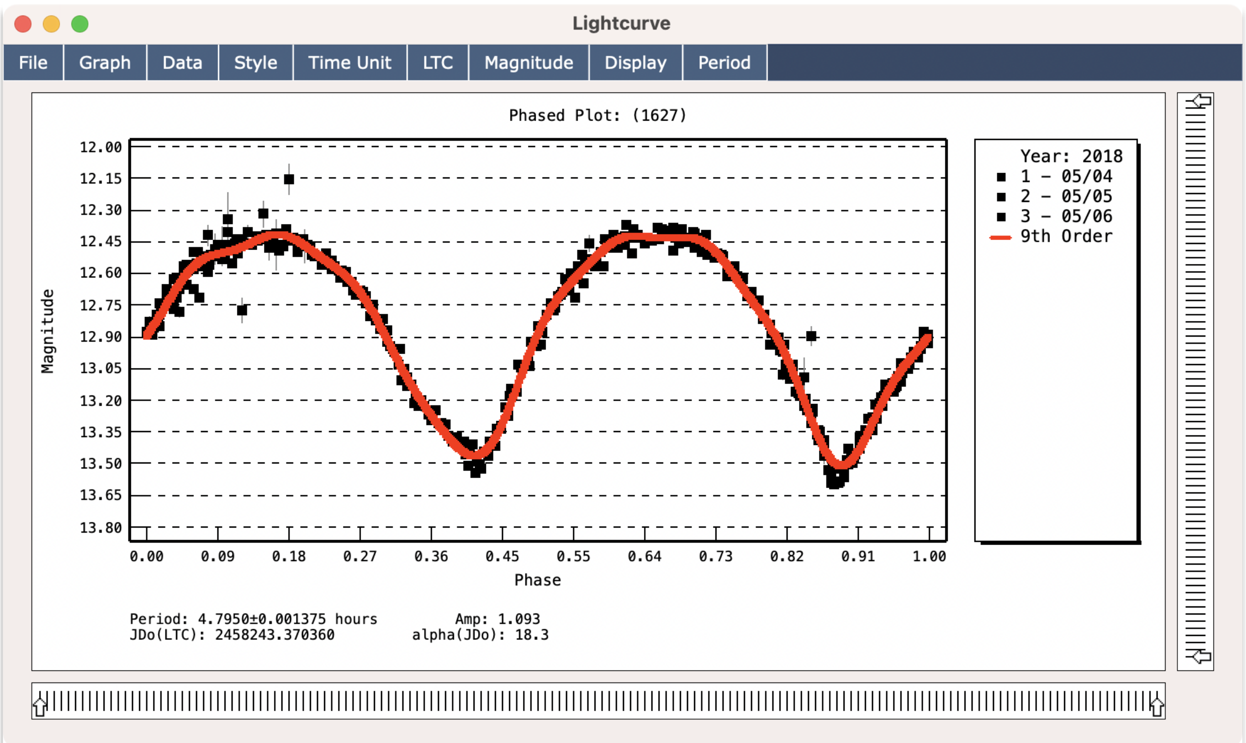


Figure 92 - Phased Plot using a 9th Order Fit

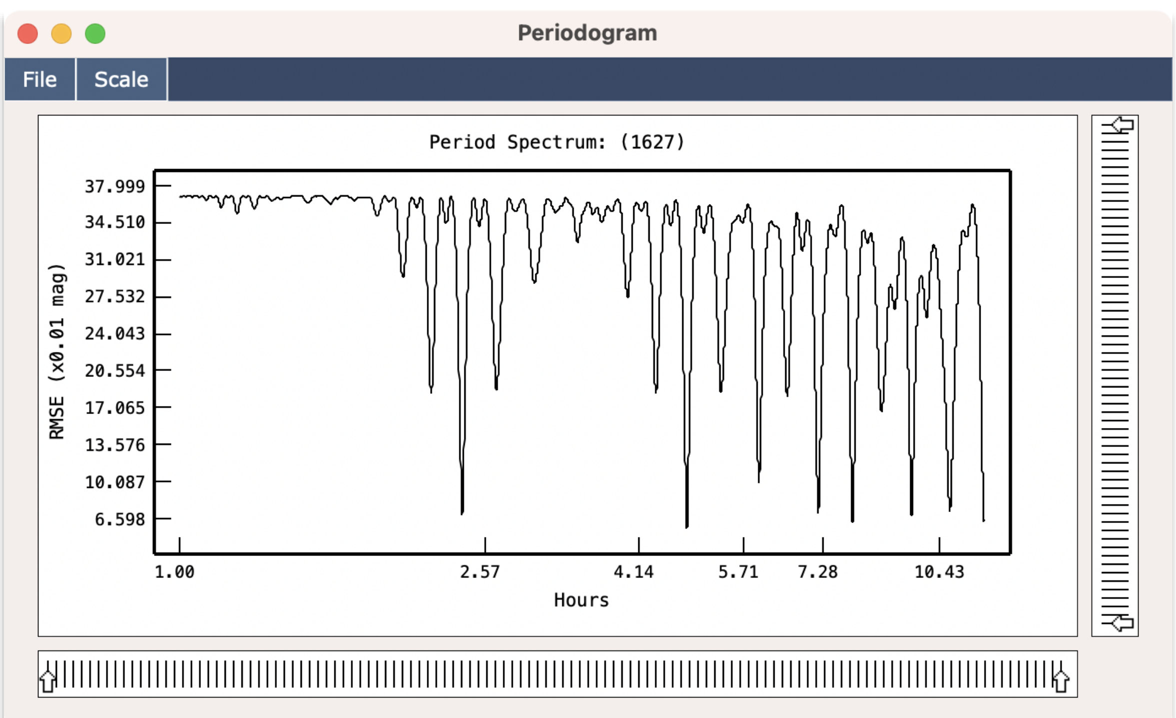


Figure 93 - Periodogram of (1627) with Search Interval of 1-12 Hours

You can also choose different styles for the graph. As an experiment, navigate to ***Style->Classic*** from the "Lightcurve" window.

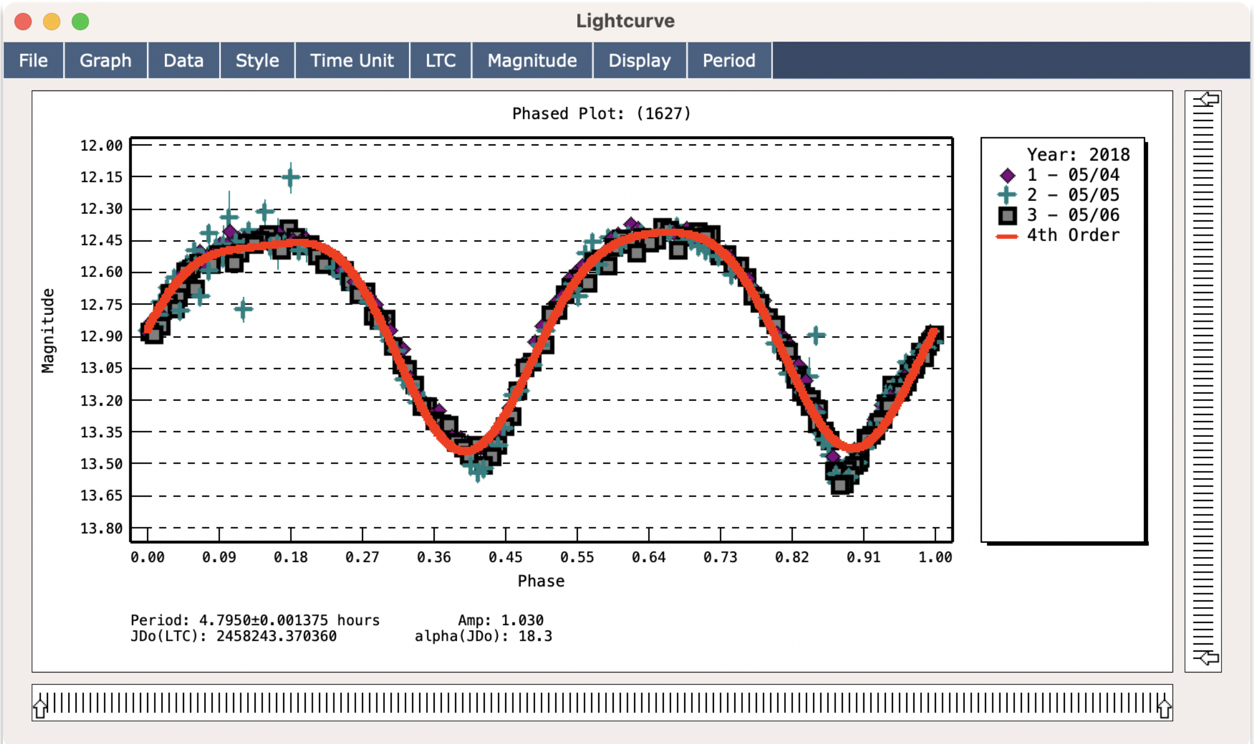
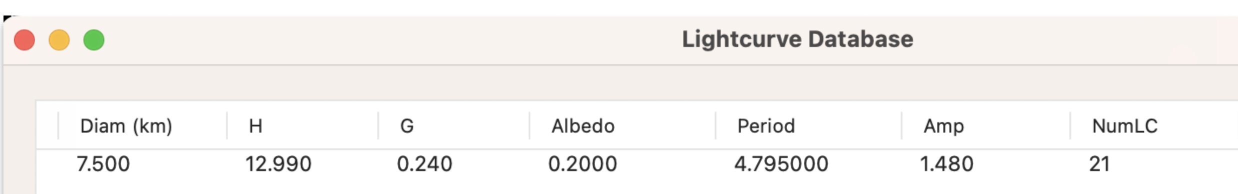


Figure 94 - Phased Plot using the Classic Style

Step 10: Compare with Lightcurve Database

As this is a known asteroid with several published rotation periods, it should be possible to determine if the period you identified is correct. To do this, navigate to ***Tools->Lightcurve Database*** from the main menu. Then, type "1627" (without the quotes) into the field next to "Search for", and also check the box labeled "Whole word only". Then click the "Search" button. It may take a moment as it has to access the online database.

When the search has completed, there should be one result returned. From this result you can determine several characteristics of the object, including its diameter, absolute magnitude (H), slope (G), albedo, period, and amplitude. From this result it should indicate a period of 4.795 hours, matching the photometry result that was generated earlier.

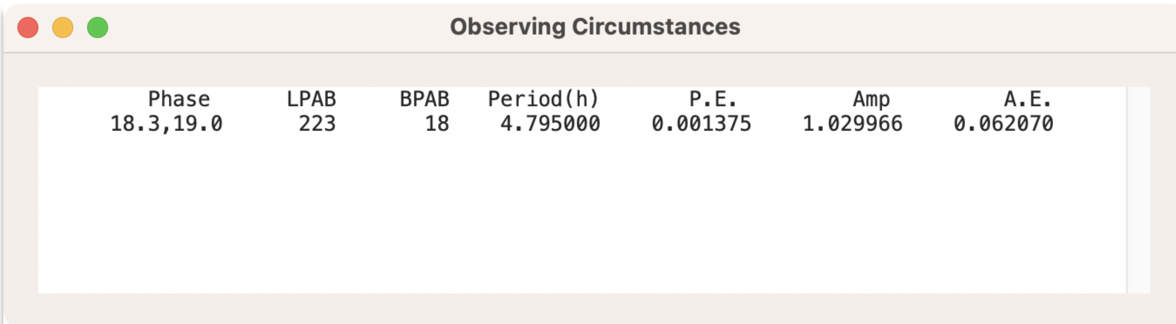


Diam (km)	H	G	Albedo	Period	Amp	NumLC
7.500	12.990	0.240	0.2000	4.795000	1.480	21

Figure 95 - Searching the Lightcurve Database

Step 11: Generate Observing Circumstances Table

If you wish to publish your results in the Minor Planet Bulletin, you may need to generate the values for the “Observing Circumstances” table. New in v9.0.6 is the ability to do so directly from the “Lightcurve” window.


Observing Circumstances

Phase	LPAB	BPAB	Period(h)	P.E.	Amp	A.E.
18.3,19.0	223	18	4.795000	0.001375	1.029966	0.062070

Figure 96 - Observing Circumstances Table Values

As shown in Figure 96, the phase, PABL, PABB, and other relevant values for the observing circumstances table are computed and displayed for publication. To access this window, navigate to ***File->Obs. Circumstances*** from the “Lightcurve” window. Note that you need to have conducted a period search in order for the period, period error (P.E.), and other associated values to be computed.

Step 12: Exporting ALCDEF Data

At some point you will probably want to share your photometry data with others. The ALCDEF format provides a standard way to do so. Navigate to ***File->Export ALCDEF*** from the “Photometry Sets” window and you will be shown a window prompting for details on the observations. When ready, click the “OK” button and the data will be exported to a text file. Alternatively, you could also export the data via ***File->Export ALCDEF (use current metadata)***. This option makes use of the metadata attached to each photometry set, which you will want to verify before using. To verify the metadata, select a photometry set and then right-click and choose “View/Edit Metadata”.

Also note that any orbit computation will be associated with that of the observatory referred to by the “MPCCode” column. If the code is 247 (indicating roving observer), then such orbit computations will use the “ObsLon” and “ObsLat” columns. These columns are populated automatically when creating photometry sets from observations, but should you need to change them, you can edit the metadata by choosing “View/Edit Metadata” from the context menu that appears when right-clicking the sets. The relevant keywords are shown in Table 1.

Table 1 - Column to ALCDEF Keyword

Column	ALCDEF Keyword
MPCCode	MPCCODE
ObsLon	OBSLONGITUDE
ObsLat	OBSLATITUDE

For more information on the ALCDEF keywords, and the ALCDEF format in particular, visit www.alcdef.org.

Step 13: Importing ALCDEF Data

To import ALCDEF data, navigate to ***File->Import ALCDEF*** from the “Photometry Sets” window. If the “Photometry Sets” window is not open, you can access it by navigating to ***File->Photometry Analysis*** from the main menu.

Once you have imported photometry, you will note that it has no object data attached to it. To apply object data, select the relevant sets to be updated and proceed to either right-click and choose “Apply Object Data”, or navigate to an existing orbit window and choose ***Ephemeris->Attach to Photometry***. Refer to Figure 88 for more details. Once object data has been applied, you will then be able to use various attributes such as Absolute Magnitude, PABL, PABB, and phase.

Adjusting Magnitude Offsets

New in v9 of Tycho is the ability to more easily adjust the magnitude offsets associated with each photometry set. Magnitude offsets can sometimes be required when working with data taken several nights apart. To experiment with this capability, load two or more photometry sets into the “Photometry Sets” window (see previous section) and then choose ***Graph->Plot All Sets*** from the menu.

Hold down the left mouse button and then drag a rectangle around the data points to be adjusted. Upon releasing the left mouse button, the enclosed data points inside the rectangle will be highlighted with a red circle. Next, right-click inside the plot area and a popup menu will appear. From this menu you can choose to adjust the magnitude offset in increments of 0.1, 0.01, and 0.001 magnitude. Alternatively, you can instead use the keyboard shortcuts: key up, key down, along with holding down the SHIFT key (0.01 increment) or CTRL key (0.001 increment). Refer to Figure 97.

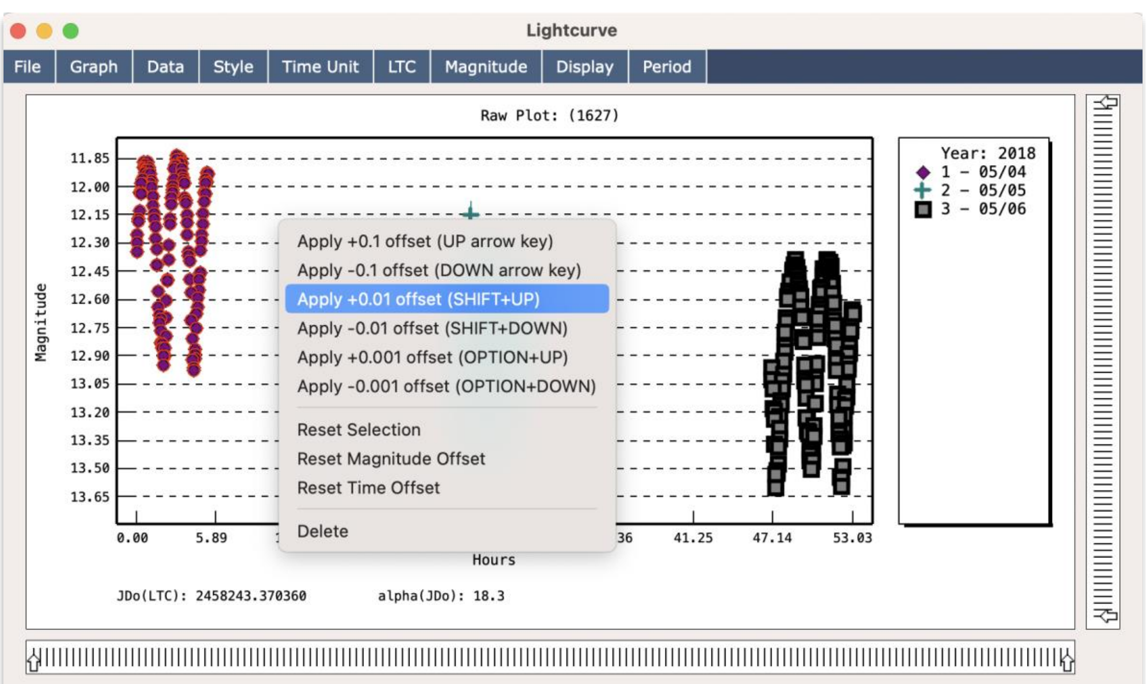


Figure 97 - Adjusting Magnitude Offsets

Generate Transformation Coefficients

The Tycho software can be used to generate transformation coefficients. These are useful if you wish to transform photometry data to the "Standard" system. The primary motivation in doing so would be to compensate for differences in the spectral response of equipment between observers.

First, you will need to supply a special set of images to Tycho. These images should capture a "Standard Field", such as M67 or NGC 7790. A list of standard fields is available from the AAVSO website:

https://app.aavso.org/vsd/stdfields

Furthermore, at least two different filters should be used throughout the image set. In this example, five filters are used: UBVRI. The images should be taken such that the filters are switched out from one image to the next. For example, one capture of UBVRI images, followed by another capture of UBVRI images. For the ultraviolet (U) images, you may need to use a longer exposure.

The procedure is documented as follows:

1. Choose a Standard Field. In this example, NGC 7790 is used. If you want to follow along with this example, you may download these images from the Tycho website at the following URL: https://storage.googleapis.com/tycho_data-1/2/NGC7790_iTelescope_T11.zip
2. Acquire images in each filter that you wish to use. Note that you may want to take a longer exposure for the U filter since most equipment is not as sensitive in that filter. This example uses 300 seconds for U and 60 seconds for the other filters.
3. Process the images. Start Tycho and choose ***List->Add Images*** from the "Image Manager" window. For this example, five images (one for each filter) are added to the list.
4. If the images have not yet been calibrated, choose ***Action->Calibrate Images*** from the main menu to proceed with calibration. The images in this example have already been calibrated, so this step should be skipped.
5. If calibration was performed in step 4, load the calibrated images into the "Image Manager". Tycho saves the calibrated images into a new directory with suffix of "_c", adjacent to the original directory.
6. Next, plate solve the images. Choose ***Action->Plate Solve Images*** from the main menu.
7. After plate solving, choose ***Action->Align Images*** from the main menu.
8. Load the aligned images into the "Image Manager". Tycho saves the aligned images into a new directory with suffix of "_a", adjacent to the original directory.
9. View the images. Choose ***Action->View Images*** from the main menu.
10. Open the "Standard Fields" window. Choose ***Photometry->Standard Fields*** from the menu of the "Image Viewer". Since the images have been plate solved, Tycho will automatically indicate the standard stars in the field with an overlay of yellow rectangles. See Figure 98.
11. Open the "Generate Transforms" window. Choose ***Photometry->Generate Transforms*** from the menu of the "Image Viewer". Refer to Figure 99.
12. Select the desired filter for "Use Images having filter label", and the corresponding "Catalog MagBand". Normally, these two selections will be identical, meaning that "B" filter label corresponds to B MagBand. But some users may have a different label, such as "Blue". Adapt as needed to your specific system. Once you have made the selection, click "Create Measurements". Repeat until measurements have been created for each filter.

13. View and edit magnitude pairs as needed. Choose ***Transforms->Generate from Measurements*** from the menu of the "Generate Transform Coefficients" window. Refer to Figure 100. A new window will appear with a list of measurements to the left and a plot of those measurements to the right.
    a. You should see a trendline and the relevant data points in the plot in the right half of the window. If the plot is empty, then it is possible that none of the data points satisfy the "Min SNR" constraint (default of which is 50.0). The other possibility is that there were no magnitude pairs found (such as B and V).
    b. If you see outlier data points in the plot, you can simply click and draw a rectangle around the data point(s) to form a selection. Once you have selected the outlier point(s), right-click in the plot and choose "Delete". If you accidently deleted the wrong data point(s), you can restore them by clicking the button "Restore Deleted". Note that this will restore every deleted data point.
    c. To view different coefficient plots, adjust the selection shown in the "Horizontal" and "Vertical" dropdowns at the bottom-left of the window. For example, the "Horizontal" dropdown allows one to specify a different magnitude pair (such as B-V or V-R). Meanwhile, the "Vertical" dropdown allows one to specify the coefficient for that pair. You can explore the different combinations, removing outliers as needed.
14. Once you are satisfied with the data, you can then click the button labeled "Generate Transform Coefficients". This will produce a report containing the numbers for your transformation coefficients. The text from this report can then either be copied and pasted into another view or, you can navigate to ***File->Save*** to save the report as a text file. Refer to Figure 101.

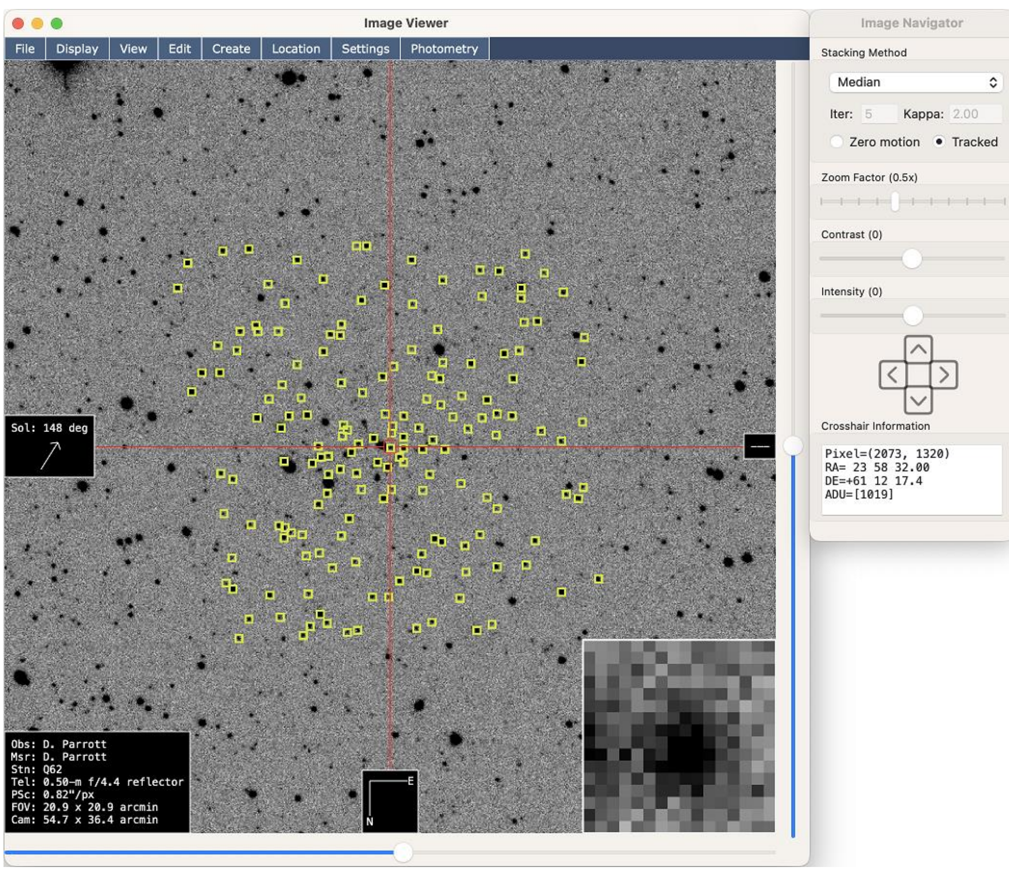


Figure 98 - Standard Field NGC 7790

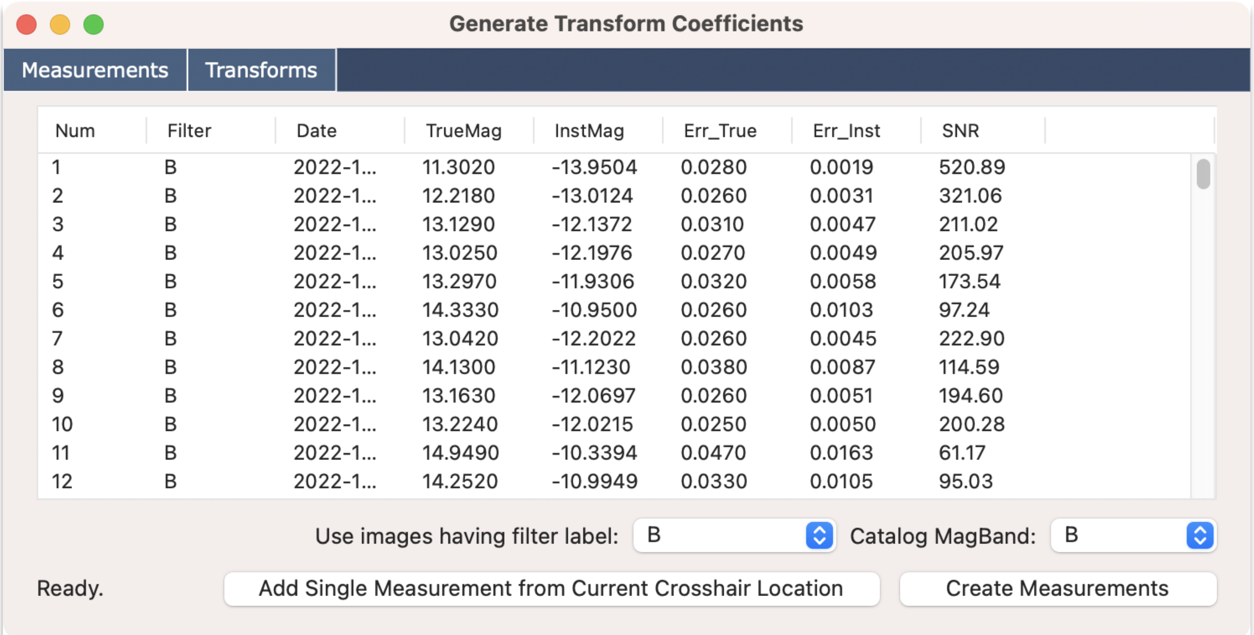


Figure 99 - Create Measurements

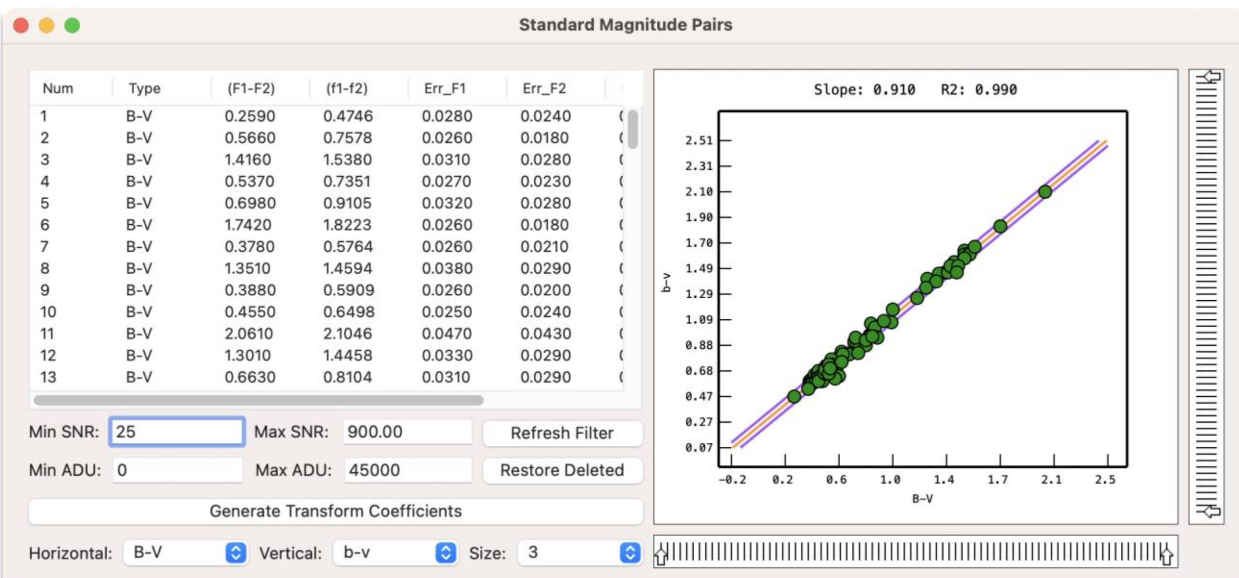


Figure 100 - Standard Magnitude Pairs

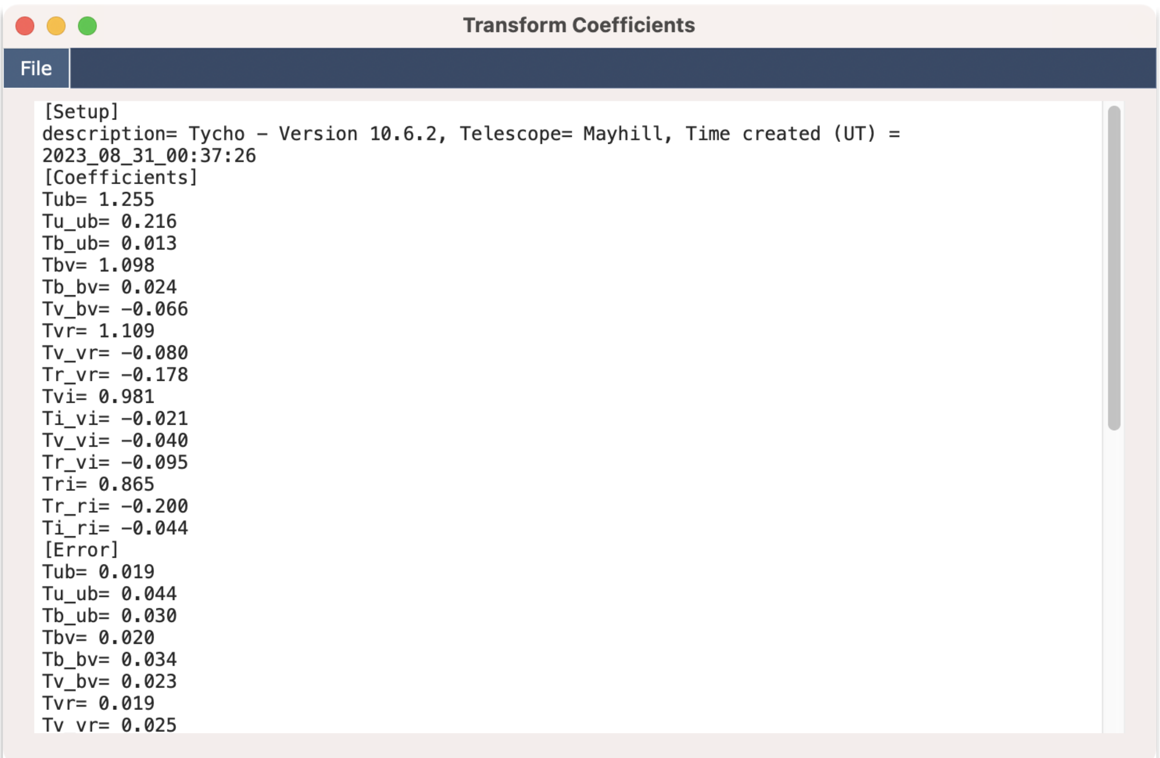


Figure 101 - Computed Transformation Coefficients

Additional notes:

For the (U-B) pair, it can be necessary to adjust the "Min SNR" filter to 25 (from the default of 50) in order to include a wider range of star colors. This can be observed by the horizontal axis. With SNR=50, it has a range from -0.5 to 0.3. But with minimum SNR lowered to 25 (allowing more stars), the horizontal axis now ranges from -0.5 to 1.2 and results in a more comprehensive trendline. Alternatively, it could have been more optimal to take a longer exposure for the U filter, as 300 seconds was not quite sufficient.

Measurements are created using the current aperture. The aperture can be modified by navigating to ***Photometry->Modify Aperture Settings*** from the "Image Viewer" menu.

While not shown in this example, it is also possible to use multiple exposures of each filter. The process would involve capturing one set of exposures for each filter, followed by another set, repeating until the desired number of sets is acquired. This can produce a trendline with more data points to work with.

Comet Photometry

New in v12 is a "Comet Photometry" module that aims to facilitate the measurement of extended objects such as comets. This functionality can be accessed by navigating to ***Photometry->Comet Photometry*** from the menu of the "Image Viewer" window.

There are several configurable items that need to be specified prior to generating measurements, as shown in Figure 103. The first three settings relate to the image data and comet position.

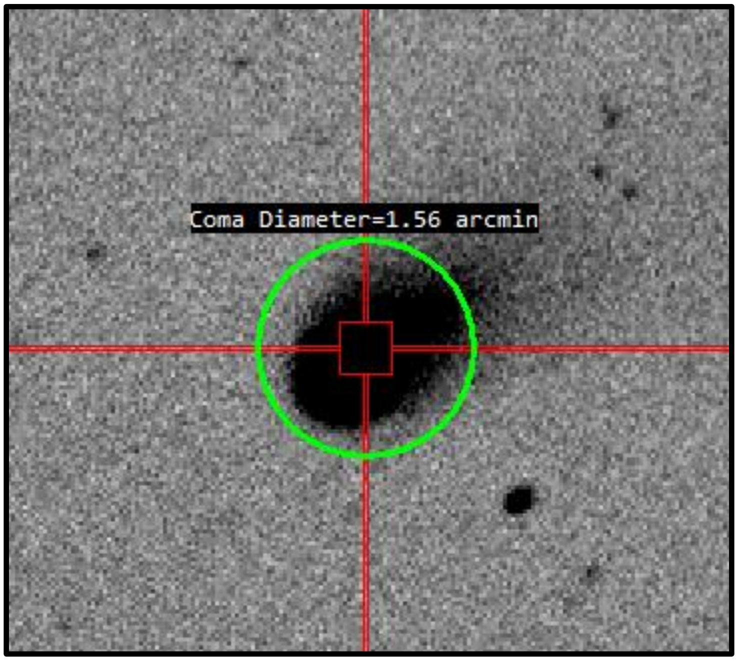


Figure 102 - Comet C/2024 G3

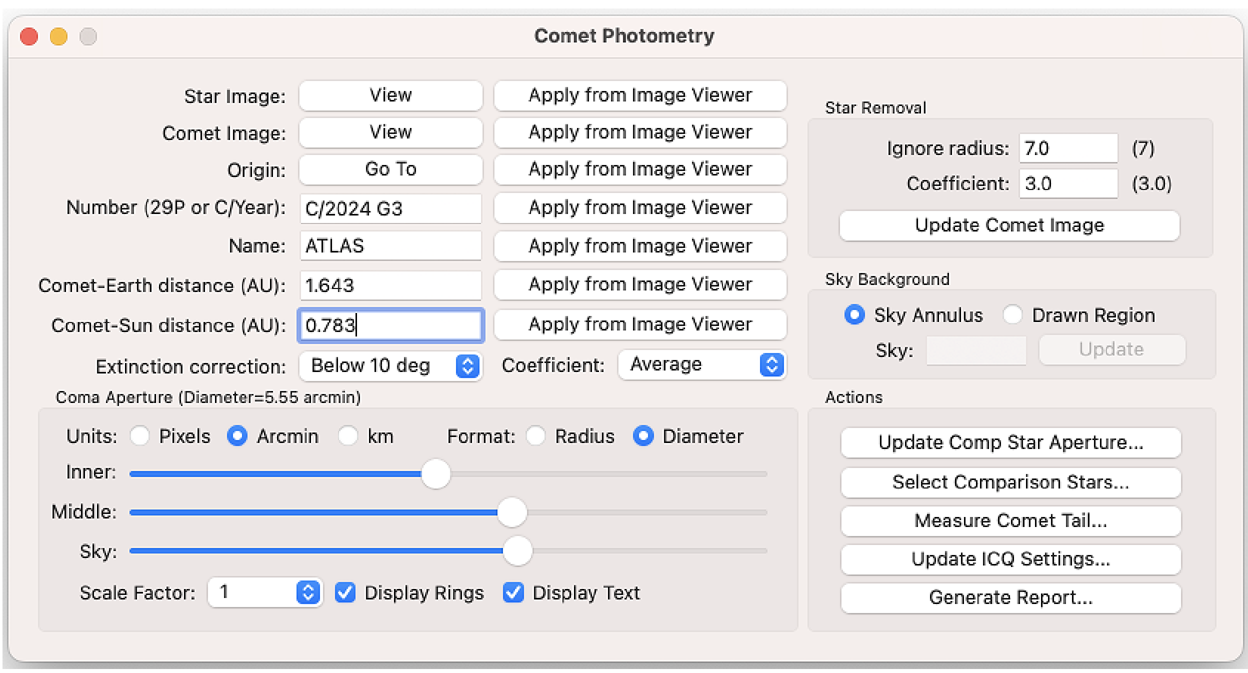


Figure 103 - Settings for Comet Photometry

Star Image: This is the image (or stack of images) that will be used to extract flux values for the comparison stars.
Comet Image: This is the image (or stack of images) that will be used to extract the comet flux.
Origin: This specifies the position of the comet as it appears on the comet image.
Number (29P or C/Year): This is the number or provisional designation of the comet. If the comet is numbered, then the number typically ends with 'P'; for example, 29P. Otherwise, if provisional

designation, it is usually in the form “C/Year XYZ”, for example, C/2024 G3. Note that sometimes the provisional designation can start with ‘P’ rather than ‘C’, as in “P/2024 S2”. This field is also automatically populated when using either the Overlay Settings or the JPL Horizons interface to attach ephemeris to the dataset.
Name: This is the comet name, such as “ATLAS” or “PANSTARRS”. If present, it will be shown in both the comet report and comet graph title.
Comet-Earth distance (AU): The distance to the comet as it appears from Earth.
Comet-Sun distance (AU): The distance to the comet as it appears from the Sun.
Coma Aperture: The measuring aperture for the comet.
Note that the parameters “Comet-Earth distance” and “Comet-Sun distance” can be populated by the following approach:

(1) Navigate to ***Settings->Overlay*** from the menu of the “Image Viewer”
(2) Click the button labeled “Apply Object” located in the lower-right corner of the “Overlay Settings” window
(3) Type the object number into the “Object Number” edit box, or alternatively the name of the object if its number is not known. Example: “13P” (without quotes) can be specified in the “Object Number” field.
(4) Click the associated “Search” button (either next to “Object Number” or “Object Name”) to begin a search for the object.
(5) Locate the returned object in the list and click “OK” to apply it to the overlay. This will also populate the comet distance values in the “Comet Photometry” settings window.

Once the comet parameters have been specified, click the “Generate Measurements” button located in the lower-right corner of the “Comet Photometry” window. Two new windows should appear, as shown in Figure 104 and Figure 105.

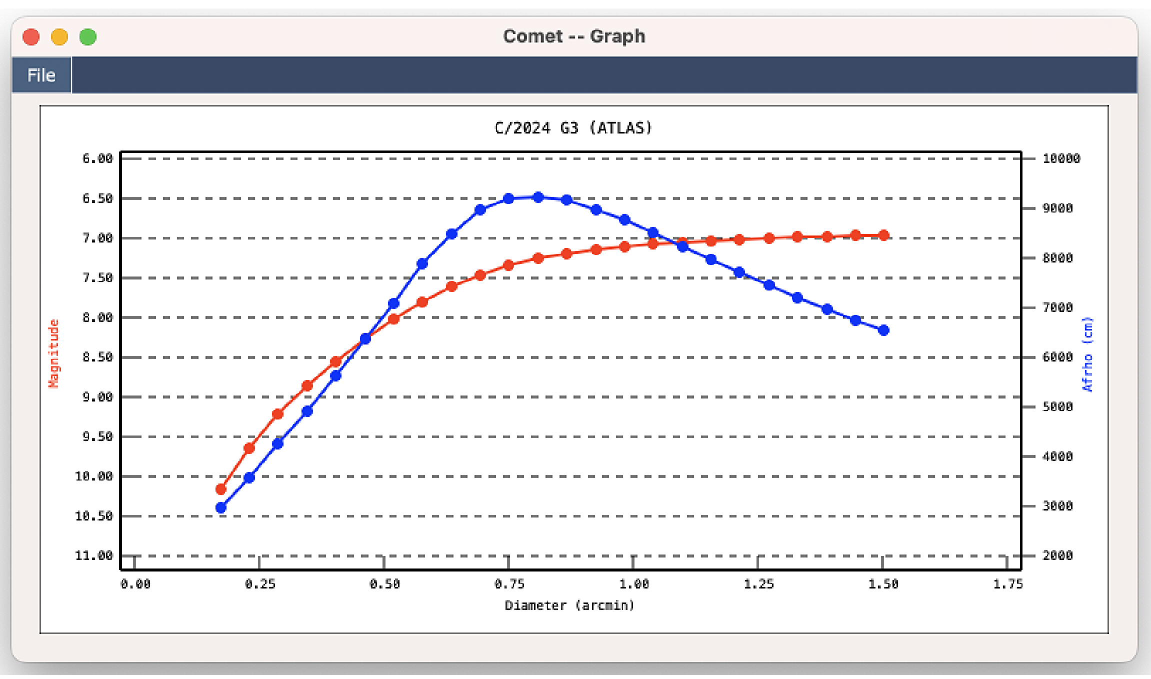


Figure 104 - Comet Photometry Graph

There is also a method by which to remove stars in the comet image. The "Ignore radius" specifies the radius at which the removal will start to take place, for example a value of 7.0 pixels indicates that image data within 7.0 pixels of the "Origin" coordinates will not be impacted. The method also operates on an adaptive threshold that is computed through the coefficient parameter. A smaller coefficient makes it more sensitive and will remove more stars, with the tradeoff that it is more likely to remove signal from the comet coma.

Note also the importance of choosing comparison stars. For large comets, it is quite possible that the automatic selection may inadvertently choose stars that are within the coma and hence result in less-than-ideal measurements. Therefore, it is advisable to manually choose the comparison stars for the comet photometry module to yield optimal results.

The comparison stars will also be measured using the same aperture settings that can be adjusted via ***Photometry->Modify Aperture Settings*** from the menu of the "Image Viewer" window. Note that for the comet itself, the target aperture is not used, since the comet photometry module will step through increasing aperture sizes. However, for the comparison stars, if the box labeled "Force comp stars to use same aperture as target" is checked, then the aperture that is specified for the target will be applied to the comparison stars. Otherwise, if it is unchecked, then the aperture that is specified in the "Comp Stars" section will be applied to the comparison stars.

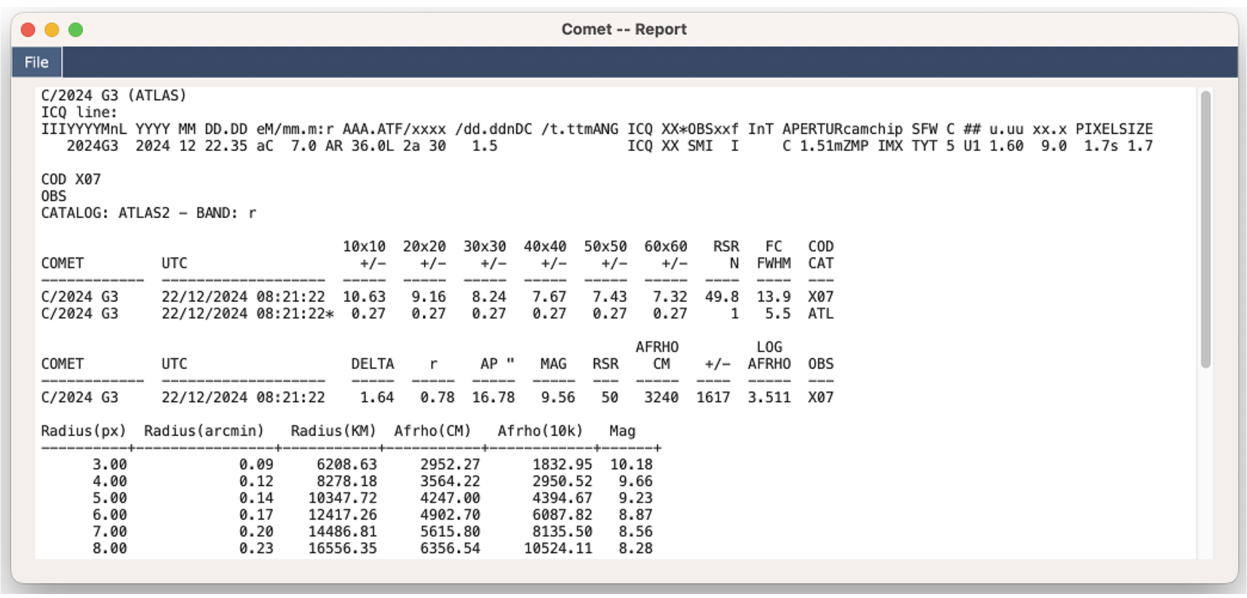


Figure 105 - Comet Photometry Report

A description of the comet report is as follows:
The first line specifies the comet. Example: C/2024 G3 (ATLAS).
After the comet name is the ICQ line. The ICQ format is described in more detail on the following page:
http://www.icq.eps.harvard.edu/ICQFormat.html

After the ICQ line is the multi-aperture report:

```
COD X07
OBS
CATALOG: ATLAS2 - BAND: r
```

COMET	UTC	10x10 +/-	20x20 +/-	30x30 +/-	40x40 +/-	50x50 +/-	60x60 +/-	RSR N	FC FWHM	COD CAT
C/2024 G3	22/12/2024 08:21:22	10.63	9.16	8.24	7.67	7.43	7.32	49.8	13.9	X07
C/2024 G3	22/12/2024 08:21:22*	0.27	0.27	0.27	0.27	0.27	0.27	1	5.5	ATL

10x10 Photometric aperture corresponding to a 10x10 arcsecond box.
+/- Standard deviation
RSR Comet signal to noise ratio.
N How many images were used to obtain the photometry.
FC Sky background magnitude.
FWHM Mean comparison star Full Width Half Minimum.
COD Observer site code
CAT Star catalogue used

Af(rho) calculation. This is done in a projected physical aperture at the comet of 10000 km. The magnitude is interpolated to this physical aperture size from the apertures measured.

COMET	UTC	DELTA	r	AP "	MAG	RSR	AFRHO CM	+/-	LOG AFRHO	OBS
C/2024 G3	22/12/2024 08:21:22	1.64	0.78	16.78	9.56	50	3240	1617	3.511	X07

DELTA Geocentric distance.
r Heliocentric distance.
AP" Equivalent aperture on the sky that gives a physical aperture of 10000 Km at the comet.
Mag Comet magnitude in the aperture.
RSR Comet signal to noise.
Afrho In cm.
+/- Error on Afrho.
Log Afrho
OBS Observer code.

A number of data points (measurements) are listed after the multi-aperture report. These measurements correspond to the data points shown on the "Comet – Graph" window. They are typically shown at every one or two pixels in increasing aperture.

Submitting to COBS

New in v13.0 of Tycho is the ability to directly the submit the Comet Photometry report to the Comet Observation database (COBS):

1. Navigate to ***File->Send to COBS*** from the menu of the "Comet -- Report" window
2. Specify the Submission Mode: Test or Normal. It is recommend to try "Test" before the actual submission.
3. Click "Send Observation".

Note: If you have not yet installed the "Send to COBS" plugin, you can do so as follows:

1. Navigate to ***Plugins->Plugin Manager*** from the main menu.
2. Choose ***Action->Download Plugin*** from the menu of the "Plugin Manager" window.

3. Right-click the "Send to COBS" entry and choose Install
4. Restart Tycho

Session Planner

Another new feature in v9 of Tycho is the Session Planner module. This module permits one to easily identify when a given object will be visible and its corresponding location in the sky. But more interesting is that it also works with both Find_Orb as well as the JPL Horizons interface. As one example, you could generate a session plan to determine when and where to look for the James Webb Space Telescope (JWST) from your particular observing location.

To activate the Session Planner, navigate to ***Tools->Session Planner*** from the main menu. Note that you may need to have the Pro license for this feature to work. Next, navigate to ***Tools->JPL Horizons Interface*** from the main menu. This will bring up the JPL Horizons interface, refer to Figure 106. Here you can specify a spacecraft (such as JWST), an asteroid or comet, or any other object given by its Two-Line Element (TLE).

In this example, we want to generate ephemeris for the James Webb Space Telescope. So, choose "Major-body (including spacecraft)" from the dropdown menu located next to "Lookup Type". Then input the value "-170" (without quotes) into the Target ID field. If you are unsure of the identifier associated with the major body that you wish to work with, you can navigate to ***Search->Major-body Lookup*** from the JPL Horizons Interface menu. Refer to Figure 107.

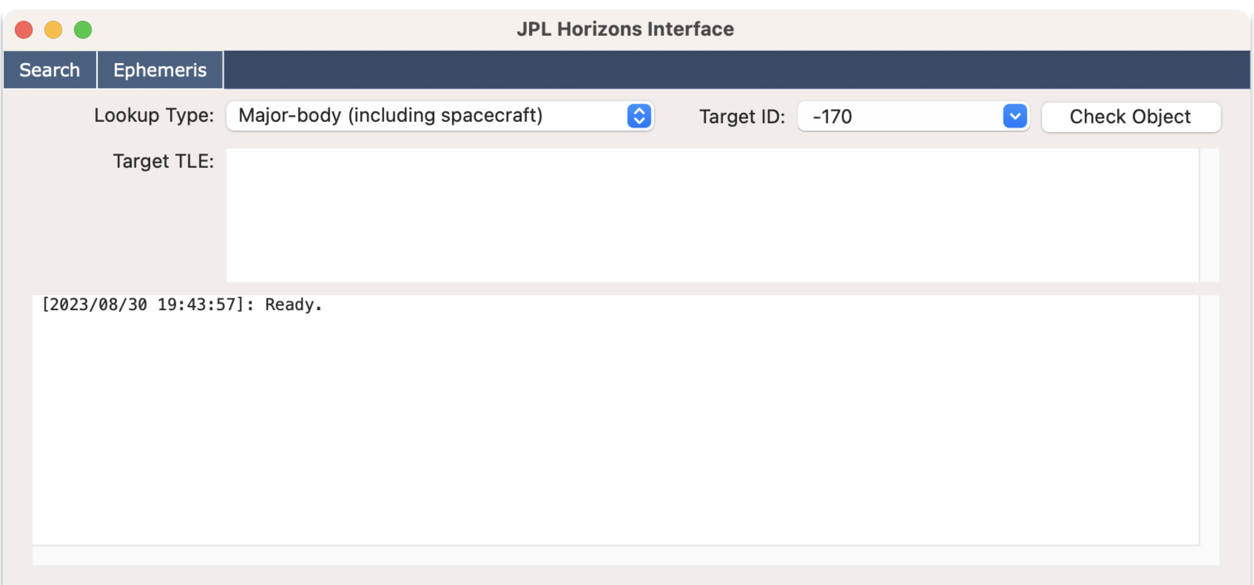


Figure 106 - JPL Horizons Interface

This will bring up a new window initially populated with every single major-body listed (607 at the time of this writing). To narrow it down to the object of interest, you can filter by "Name contains" or "Alias contains" in the two edit boxes at the bottom of the window. For example, you could type "James" (without quotes) into the "Name contains" field and it would reduce the list to just that of the James Webb Space Telescope. You will then notice that the "Ident" column is equal to -170 for this particular object. Now that you have the identifier, you can specify it in the “Target ID” field in the interface window.

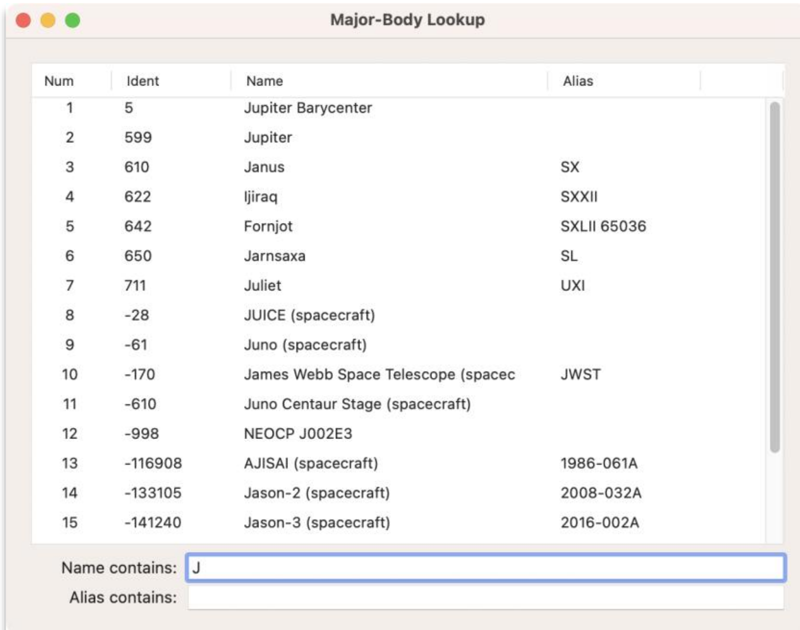


Figure 107 - Major-body Lookup for JPL Horizons Interface

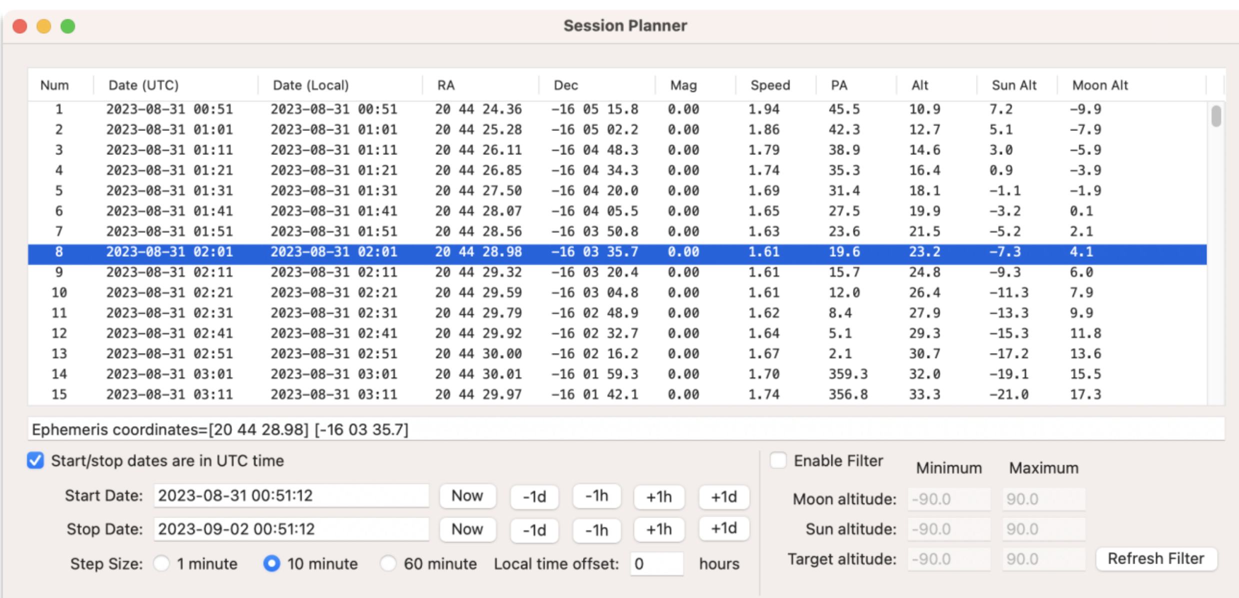


Figure 108 - Session Planner

At this point, you have specified the “Lookup Type” to be that of Major-body, and the “Target ID” to be that of -170. You would next proceed to navigate to ***Ephemeris->Attach to Session Planner*** from the “JPL Horizons Interface” menu, but before doing so, you will want to update the “Start Date”, “Stop

Date", and "Step Size" fields located at the bottom of the Session Planner window. Refer to Figure 108 for an example.

Here you can see that the "Start Date" is August 31, 2023 and the "Stop Date" is September 2, 2023. The "Step Size" is 10 minutes. The dates are also in UTC time as indicated by the checkbox. If you want, you can also specify the "Local time offset", associated with your observing location. If you unchecked the box indicating that the dates are in UTC time, then that offset will be applied to the start and stop dates for the request. It will also be shown in the "Date (Local)" column. Note that the time offset does not take into account Daylight Savings Time (DST) – it is just a simple time offset for convenience purposes.

Once you are satisfied with the "Start Date", "Stop Date", and "Step Size" parameters, navigate back to the JPL Horizons Interface window and proceed to choose ***Ephemeris->Attach to Session Planner*** from its menu. If all goes well, your planner window should resemble that as shown in Figure 108, with the list showing the coordinates of the object for each particular time and date. You can also choose to enable a filter to reduce the results such that the list will only show coordinates whenever the object is above a certain altitude, and/or whenever the moon and sun are below a certain altitude.

The Session Planner also works with orbit computation windows, so if you have an object that is not available from the JPL Horizons Interface (perhaps a newly-discovered object), then you can still generate a plan by choosing ***Ephemeris->Attach to Session Planner*** from respective orbit window.

Command Line Interface

Various parts of the Tycho functionality can be accessed via the command line. The following presents a list of such functions. Future versions may also expand upon this list.

On Windows systems, a sample invocation may resemble the following:

"C:\Program Files\Tycho\Tycho.exe" 1 "C:\Users\Daniel\Desktop\data\ds3"

On macOS systems, a sample invocation may resemble the following:

open -n "/Applications/Tycho.app" --args 1 "/Users/daniel/Desktop/data/ds3"

AutoRun

```
AutoRun requires three command line arguments, with an optional fourth argument
Usage: <tycho> <mode> <path to image directory> [override file]
<tycho> = Full path to Tycho executable
<mode>  = 1 (for regular autorun) or 101 (for image subset autorun)
<path>  = path to input directory
[over]  = path to override file (will use 'override.txt' if not specified)
```

Image Preview

```
Image preview requires three arguments
Usage: <tycho> <mode> <path>
<tycho> = Full path to Tycho executable
<mode>  = 3
<path>  = path to image
```

Debayer

```
Debayer requires 8 arguments
Usage: <tycho> <mode> <cpu> <pattern> <output> <mode> <input dir> <output dir>
<tycho> = tycho executable
<mode>  = 12
<cpu>         0=use existing settings, 1=force CPU mode
<pattern>     0=auto,1=RGGB,2=BGGR,3=GBRG,4=GRBG,5=GRGB,6=GBGR,7=RGBG,8=BGRG
<output>      0=Combined Luminance 1=Split RGB
<mode>        0=Superpixel 1=Bilinear 2=VNG
<input dir>
<output dir>
```

Calibration

```
Calibration requires 9 arguments
Usage: <tycho> <mode> <cpu> <dark_frame> <flat_frame> <normalize> <fix_pixels> <input
dir> <output dir>
<tycho> = Full path to Tycho executable
<mode>  = 7
<cpu>         0=use existing settings, 1=force CPU mode
<dark_frame>  0=no dark, is file path=use that file, path is dir=select from dir
<flat_frame>  0=no flat, 1=pseudo, is file path=use file, path is dir=select from dir
<normalize>   0=no normalization, 1=perform normalization
<fix_pixels>  0=do not fix hot pixels, 1=fix hot pixels
<input dir>
<output dir>
```

Resize

```
Resize requires 14 arguments
Usage: <tycho> <mode> <cpu> <width> <height> <tcrop> <rcrop> <bcrop> <lcrop> <bin> <hdiv>
<vdiv> <input dir> <output dir>
<tycho> = Full path to Tycho executable
<mode>  = 8
<cpu>    0=use existing settings, 1=force CPU mode
<width>  0=keep width unchanged; otherwise, specifies desired width
<height> 0=keep height unchanged; otherwise, specifies desired height
<tcrop>  0=no cropping at top; otherwise, specifies desired top cropping
<rcrop>  0=no cropping at right; otherwise, specifies desired right cropping
<bcrop>  0=no cropping at bottom; otherwise, specifies desired bottom cropping
<lcrop>  0=no cropping at left; otherwise, specifies desired left cropping
<bin>    1=1x1 (no binning), 2=2x2 binning, etc
<hdiv>   1=no divisions, 2=divide by 2 in horizontal, etc
<vdiv>   1=no divisions, 2=divide by 2 in vertical, etc
<input dir>
<output dir>
Note: Desired dimension overrides crop amount.
```

Align

```
Alignment requires 9 arguments, with an optional reference index
Usage: <tycho> <mode> <cpu> <num_threads> <interpolate> <align_mode> <dist_corr>
[idx_ref] <input dir> <output dir>
<tycho> = Full path to Tycho executable
<mode>  = 9
<cpu>         0=use existing settings, 1=force CPU mode
<num_threads> = Number of threads to use for the alignment process
<interpolate> 0=bilinear, 1=bicubic, 2=lanczos-3
```

```
<align_mode>  0=Star Match, 1=WCS Match
<dist_corr>   = Whether or not to apply distortion correction (0=no, 1=yes)
[idx_ref]     = Index (range of 0 to N-1) of the reference image (optional)
<input dir>
<output dir>
```

Merge

```
Merge requires 6 arguments
Usage: <tycho> <mode> <num_threads> <max_delta_time> <input dir> <output dir>
<tycho> = Full path to Tycho executable
<mode>  = 10
<num_threads> = Number of threads to use for the merge process
<max_delta_time> = Max amount of time (in seconds) between images to be considered a
match
<input dir>
<output dir>
```

Cross-Match

```
Cross-match requires 5 arguments
Usage: <tycho> <mode> <create_snapshots> <path_a> <path_b>
<tycho> = Full path to Tycho executable
<mode>  = 11
<create_snapshots> 0=do not create snapshot images; 1=create snapshot images
<path_a> = Full path to the root directory of the 'A' camera track files
<path_b> = Full path to the root directory of the 'B' camera track files
```

Object Linker

Object Linker interface requires 7 arguments

Usage: <tycho> <mode> <link_timeout> <num_threads> <dirs_set_a> <dirs_set_b> <path_out>

```
<tycho> = Full path to Tycho executable
<mode>  = 14
<link_timeout> = Timeout, in seconds, for each link
<num_threads> = Number of threads to operate during linkage
<dirs_set_a> = Full path to file listing the directories for Set 'A'
<dirs_set_b> = Full path to file listing the directories for Set 'B'
<path_out> = Full path to output linker result
```

Star Extractor

Star Extractor interface requires 6 arguments

Usage: <tycho> <mode> <downsample> <extract_mode> <path_in> <path_out>

```
<tycho> = Full path to Tycho executable
<mode>  = 15
<downsample> = Downsample, from 1 to 10 (default of 2).
<extract_mode> = 1=Standard, 2=Extended
<path_in> = Full path to the directory containing subdirectories of .fits image files
<path_out> = Full path to the output directory
```

Drizzle

Drizzle requires 10 arguments

Usage:

<tycho> <mode> <cpu> <scale> <mode> <method> <ref img> <drop size> <input dir> <output dir>

```
<tycho> = Full path to Tycho executable
<mode>  = 16
<cpu>   0=use existing settings, 1=force CPU mode
<scale> 1=1x scale 2=2x scale
<mode>  0=Per Filter 1=Combine All
<method> 0=Average   1=Median
<ref img> 0=Auto  1=First  2=Middle  3=Last
<drop size>
<input dir>
<output dir>
```

Lua Script

Lua script requires 5 arguments

Usage:

<tycho> <mode> <bkgnd> <path_script> <input dir>

```
<tycho> = Full path to Tycho executable
<mode>  = 17
<bkgnd> = 0-not bkgnd operation  1-run in bkgnd (hide window)
<path_script>
<input dir>
```

Load Known Objects in Image

Load Known Objects in Image requires 4 arguments

Usage:

<tycho> <mode> <input dir> <output csv file>

```
<tycho> = Full path to Tycho executable
<mode>  = 18
<input dir>
<output csv file>
```

Monitor Queue

New in v12 is a feature called "Monitor Queue". This allows Tycho to watch a directory for incoming datasets. One application of this is to have Tycho process new datasets in an automated fashion. To launch the Monitor Queue, navigate to ***File->Monitor Queue*** from the main menu.

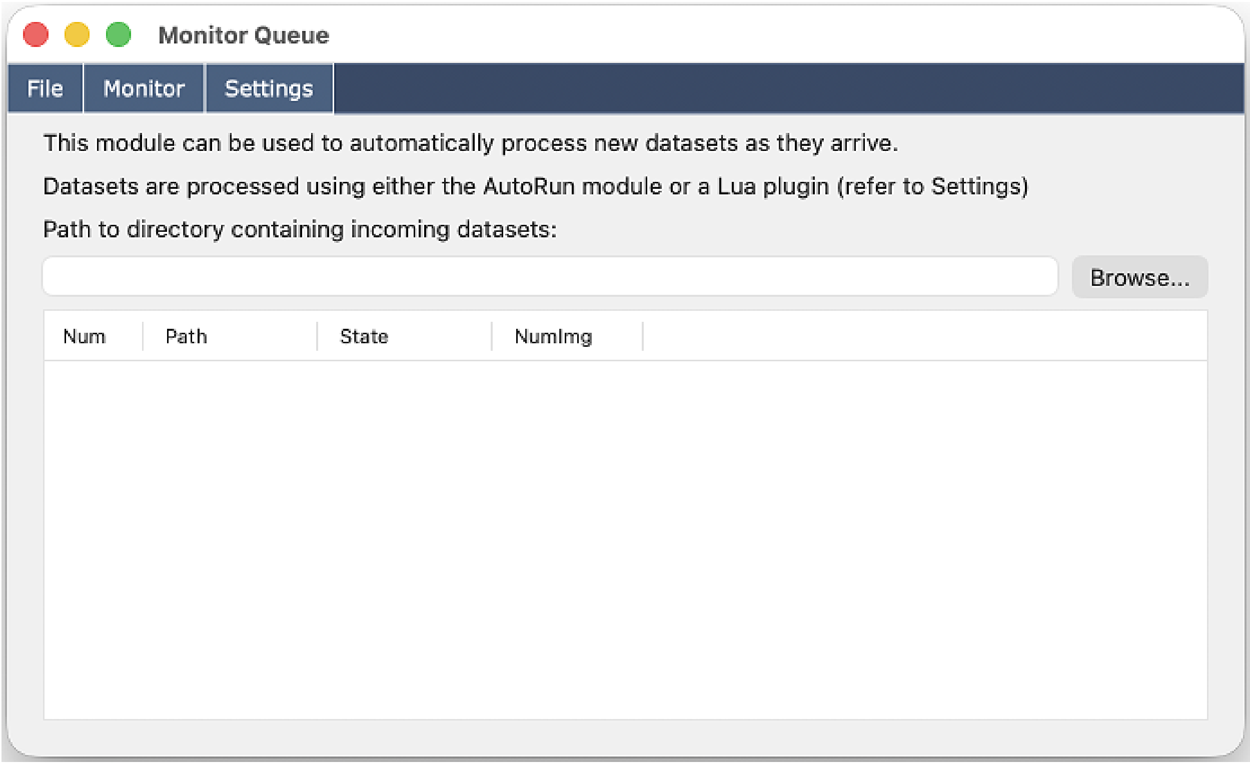


Figure 109 - Monitor Queue

As shown in Figure 109, the first setting is to specify the directory to be watched. Any new datasets that arrive inside this directory after starting the monitor will be processed according to the Auto Run settings. Refer to the section "Auto Run (Scripting)" for more details on how to configure Auto Run.

Configuring the Monitor Queue

There are a few settings that can be adjusted for the Monitor Queue. Navigate to ***Settings->Configure…*** from the menu of the "Monitor Queue" window.

"Processing Mode": Choose either "AutoRun" or "Lua script". If choosing Lua script, specify the path to the entry point of the script (main.lua, as an example).

"Dataset Ready": This indicates the criteria in which the Monitor Queue will deem a dataset ready for processing. The default value is 5 seconds. This means that if 5 seconds have elapsed with no change in the number of images located in the dataset directory, that dataset is considered ready for processing, unless it does not meet a specified minimum image count.

“Ignore datasets having fewer than”: This setting allows the Monitor Queue to require datasets to have a certain minimum number of images. Datasets not meeting this minimum will not be marked ready even if the appropriate time has elapsed in which there has been no change in image count.

“Split datasets”: This option allows the Monitor Queue to split datasets into smaller sized datasets whenever a specified image count is reached.

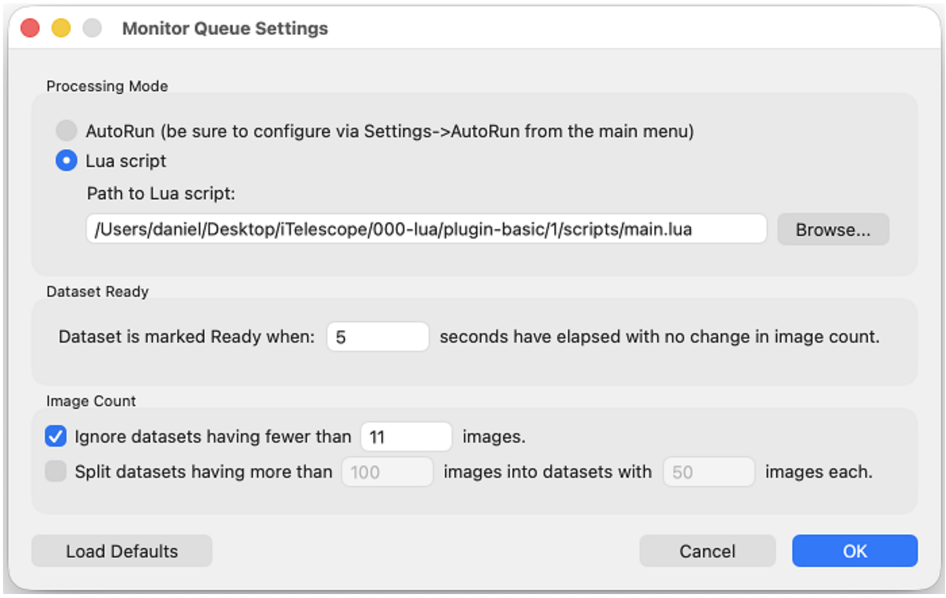


Figure 110 - Monitor Queue Settings

Start the Monitor Queue

Once it has been configured, the Monitor Queue can be started to begin watching for new datasets. Navigate to ***Monitor->Start*** from the menu of the “Monitor Queue” window. **Note**: Datasets that were initialized prior to starting the queue are excluded from the monitor list, even if they receive new images. Only datasets that are initialized after starting the queue are monitored.

Stop the Monitor Queue

When finished with the Monitor Queue, navigate to ***Monitor->Stop*** from the menu of the “Monitor Queue” window. This halts the automatic processing of any new datasets.

Removing a Dataset

Sometimes it can be desirable to remove a dataset from the monitor list, either to clean up the list of already processed datasets, or in the event a dataset is stuck in “Processing” state (which may occur if there was an error in processing). To remove a dataset, right-click on it and choose “Remove” from the popup menu that appears.

Satellite Identification

There are three ways to identify artificial satellites in Tycho:

(1) From "Image Viewer"
(2) From "Fast Tracker Results"
(3) From "IOD Report" (observations)

Satellite Identification: Image Viewer

If the "Image Viewer" is not already open, proceed to open it by navigating to ***Action->View Images*** from the main menu. The assumption here is that the images have already been plate solved and aligned. If not, please review the earlier sections in the user guide on how to plate solve and align images. Once the "Image Viewer" is open, proceed to navigate to the object requiring identification.

Once the object has been centered, there are two ways to proceed with its identification:

(1) Creating a ruler to specify position and motion
(2) Using markers to set position (motion optional)

The first approach is generally preferred when the direction of the object is known. For example, if you have multiple images, you can blink through them to establish the object direction. The second approach can be useful if you have only one image and are unsure of the motion of the object (or if the image timestamps are inaccurate).

Method 1:

Right-click in the "Image Viewer" and choose "Create Ruler" from the popup menu that appears. The cursor will be surrounded by another shape indicating marker placement mode. Set the first marker by clicking at the starting location of the object streak. Next, set the second marker by clicking at the stop location of the object streak. Now that you have defined the two markers, right-click once again and choose "Create Track – Satellite ID" from the popup menu. You should see a new entry appear in a window labeled "Fast Tracker Results".

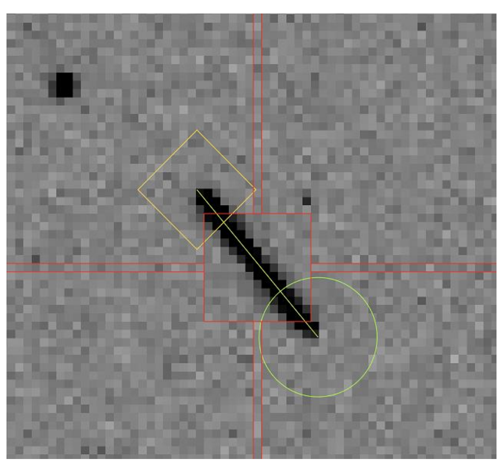


Figure 111 - Method 1: Defining Position and Motion

Method 2:

When the motion of the object is unknown, you can simply set the two markers to the midpoint of the object streak. **Important**: Be sure that the two markers are at the exact same placement – this will ensure that the motion is excluded from the matching criteria. To do this, right-click in the "Image Viewer" and choose "Create Marker 1". Then, right-click again and choose "Create Marker 2". Both markers should now be defined at the exact same placement, indicating position but no motion. Now right-click once more and choose "Create Track – Satellite ID" from the popup menu. You should see a new entry appear in a window labeled "Fast Tracker Results".

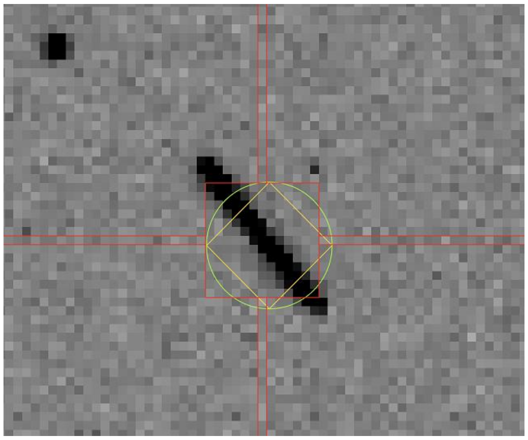


Figure 112 - Method 2: Defining Position (no motion)

Now that you have added an entry to the "Fast Tracker Results", refer to the section "Satellite Identification: Fast Tracker Results" for details on how to proceed with the identification process.

Satellite Identification: Fast Tracker Results

The "Fast Tracker Results" window typically appears after having invoked the "Fast Tracker" operation, which will return all detected streaks and other moving objects in the dataset. However, it can appear after manually adding an entry to the list (such as might occur via the "Image Viewer" or from the "IOD Report" with a set of observations). Whenever there is at least one entry in the list, you can navigate to ***File->Match ArtSats*** from the menu of the "Fast Tracker Results" window in order to invoke the identification routine. After a few seconds, the list should be updated with any returned identifications, depending on the matching criteria (see "ArtSat Match Configuration" for more details).

ObjSpeed: This is the predicted speed of the matched object.
ObjPA: This is the predicted position angle of the matched object.
ObjNum: This is the catalog number of the matched object.
ObjName: This is the catalog name of the matched object.
ObjDist: This is the cross-track distance, in arcminutes, of the matched object.
Offset(s): This is the along-track offset, in seconds, of the matched object. "L" means the object is "Late" compared to the time associated with its predicted position; "E" means "Early".

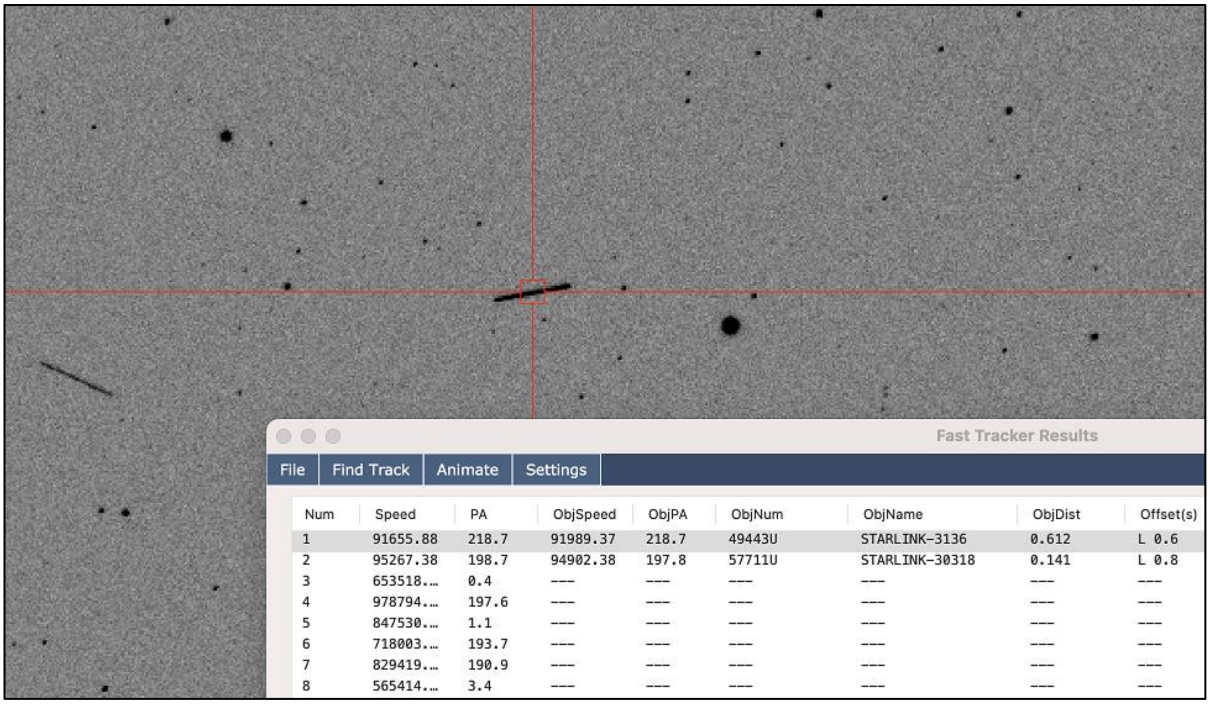


Figure 113 - Example: Two Starlink Satellites

Satellite Identification: IOD Report (Observations)

If one has only a set of observations – or even just a single observation – it can still be possible to perform satellite identification. The following formats are accepted:

(1) MPC1992
(2) ADES
(3) IOD
(4) RDE
(5) UK

Note: The accuracy of the identification depends not just on the accuracy of the observation(s) but also on the accuracy of the observer location information. Typically, the matching accuracy is higher when using the other two methods ("Image Viewer" and direct from "Fast Tracker" results), since those operate with higher precision internally compared to using external observations which have undergone more rounding/truncation operations.

(1) Navigate to ***Satellites->Observations*** from the main menu.
(2) Copy and paste the observations into the edit box.
(3) If the observations are in a format other than IOD, invoke the relevant conversion routine from the ***Convert*** menu of the "IOD Report" window.
(4) Navigate to ***TLE Analysis->Find Matching ArtSat*** from the menu of the "IOD Report" window.

Note: Supplying more than one observation can be helpful, but **only** if the additional observations are close in time (usually no more than a few seconds at most). If the observations are separated by several minutes, for example, the computed motion between them may be less accurate, and the additional observations could actually reduce the likelihood of an accurate match in that scenario.

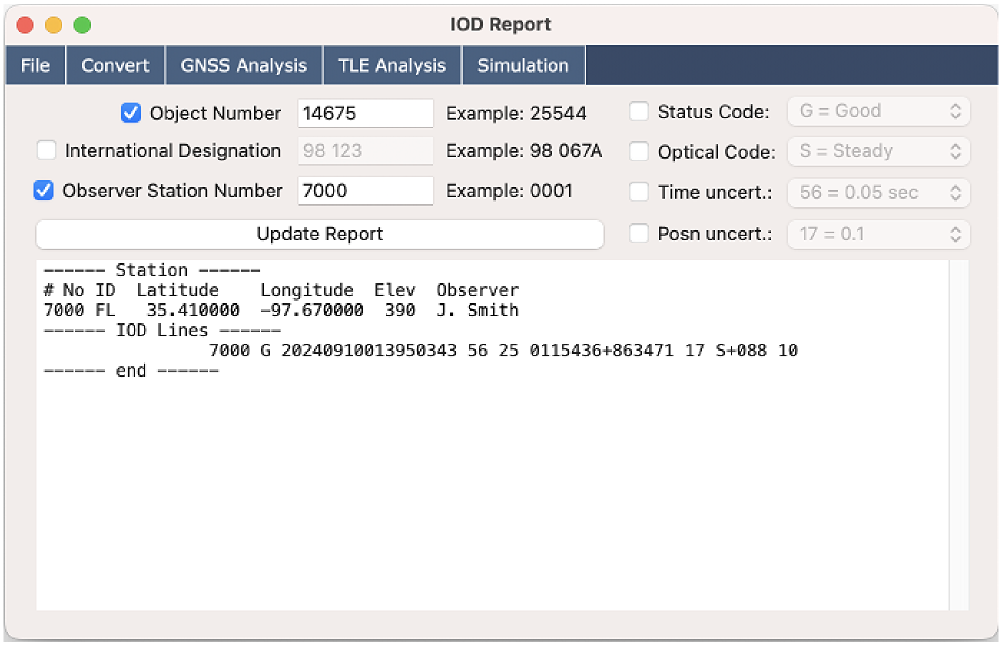


Figure 114 - IOD Report with Single Observation

Multiple Matches

The identification routine attempts to isolate the best possible match for a given object, based on motion (if available) and the cross-track and along-track offsets. If there are additional candidate matches, right-click on the track and choose “Show Top 5 Matches…” from the popup menu that appears. This will present a new window listing the top five matches.

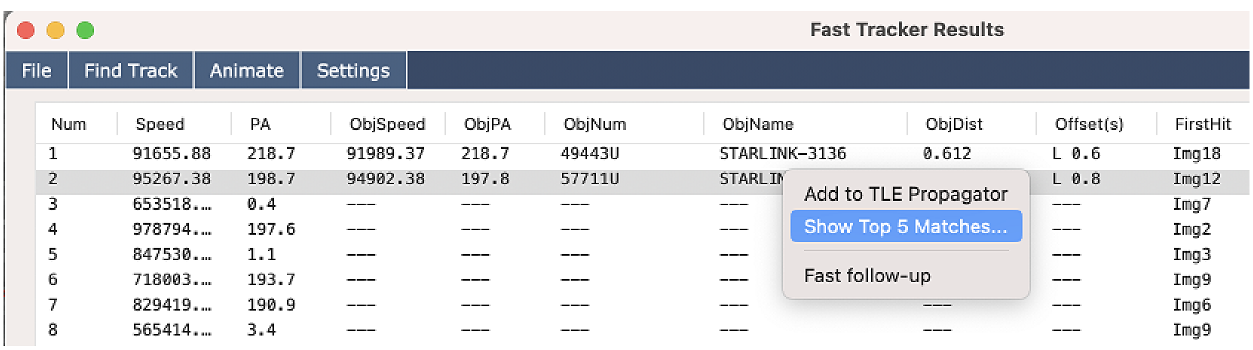


Figure 115 - Example: Showing the "Top 5 Matches"

As shown in Figure 116, the identification routine found two possible matches for the selected track. The first match has very low offsets of 0.14 arcminutes and 0.8 seconds (Late), while the other candidate match is much less likely to be the match as it has notably higher offsets of 26.51 arcminutes and 101.8 seconds (Late). Note also the position angle (PA) component of the motion: the most likely match has better agreement (197.8 degrees) with the track PA compared to that of the unlikely match.

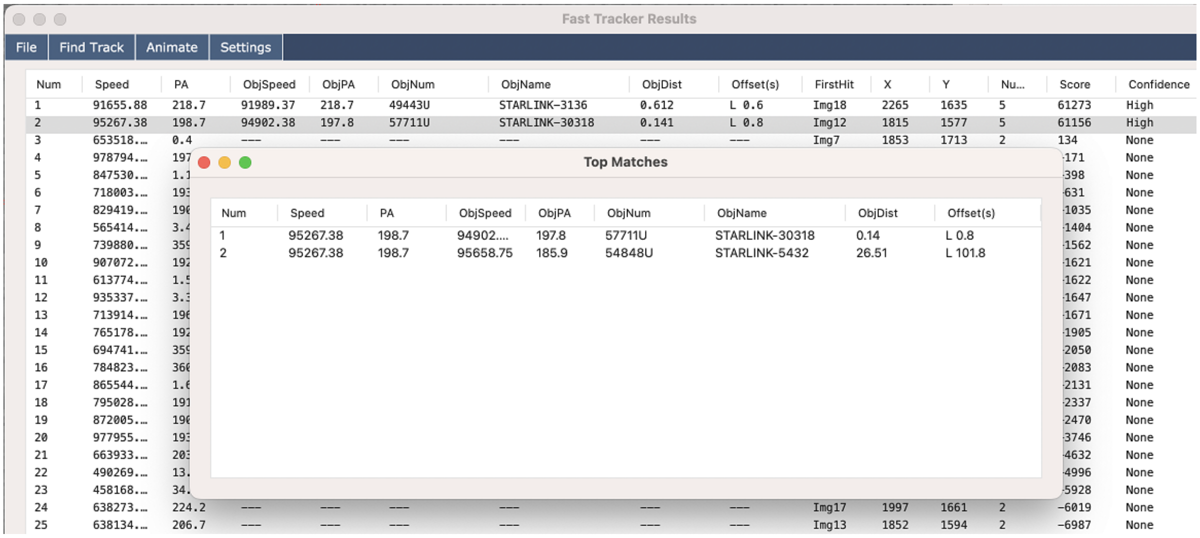


Figure 116 - Top 5 Matches

ArtSat Match Configuration

The identification routine can be configured by navigating to ***Settings->ArtSat Match Configuration*** from the menu of the "Fast Tracker Results" window.

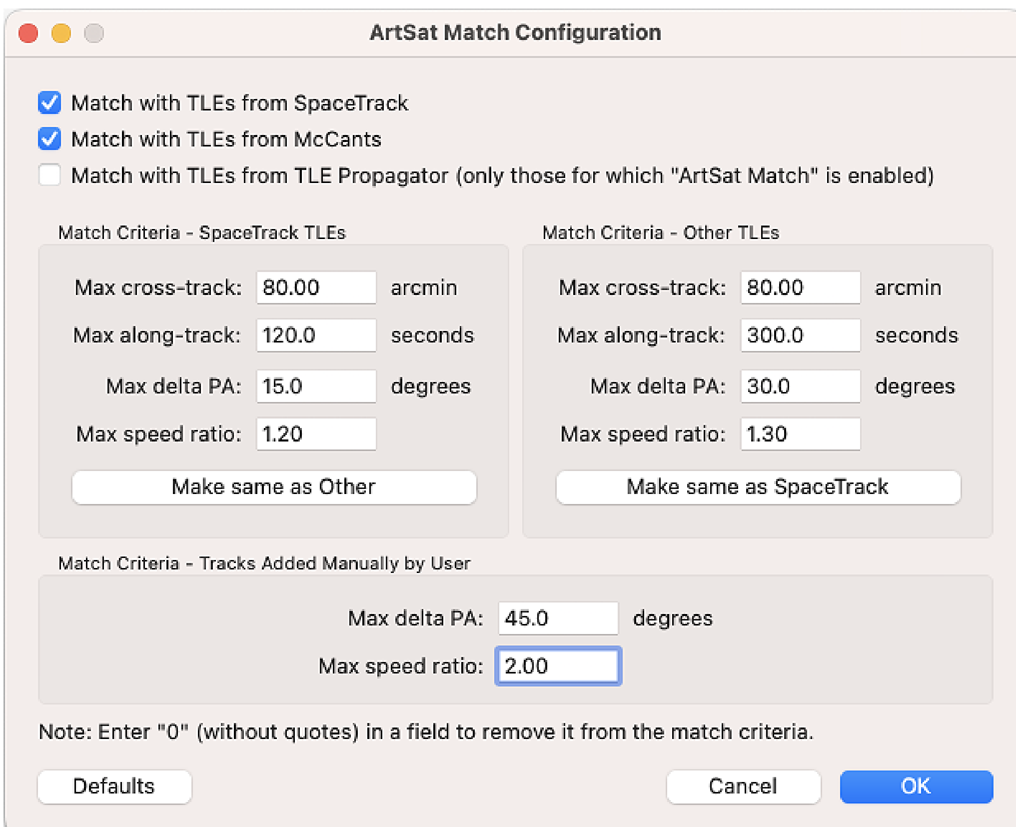


Figure 117 - ArtSat Match Configuration

From this configuration panel it is possible to specify which sources to use for the identification. The most useful source is the SpaceTrack catalog, which maintains two-line elements (TLEs) on over 23,000 objects. It is also possible to include elements from the TLE Propagator module, which stores user-defined elements.

The Match Criteria dictates the thresholds at which a candidate match can be included in the identification. The first two parameters are cross-track and along-track offsets. Objects which are less than these limits can be included for consideration in the identification. The next two parameters are motion limits: delta PA indicates the maximum difference in position angle that is permitted, while max speed ratio indicates the maximum ratio in speed that is permitted. For example, a ratio of 1.20 allows for the candidate match to have a speed that is within 20% of the computed track speed.

Tracks that are added manually (such as via the "Image Viewer" or through the IOD Report) can be given a more lenient matching criteria on the motion parameters. It is also possible to remove fields from the criteria altogether by simply inputting "0" (without quotes) in the relevant text box. For example, specifying 0 for the max speed ratio will remove the speed comparison from the matching routine.

Once you are finished making any desired changes, click the "OK" button to save the new settings. Then, navigate to ***File->Match ArtSats*** from the menu of the "Fast Tracker Results" window to see the effects of the changes.

Fast Tracker

One significant new feature in v12 is the "Fast Acquisition of Streaked Targets" (F.A.S.T.) Tracker module that enables the rapid detection of moving objects even in large datasets. The detection process also accommodates targets having high variability in brightness and can robustly isolate and track multiple objects in the same field. It can also process objects having only a few pixels of streak, being able to detect very slow-moving objects alongside extremely fast-moving objects. One application of FAST is to dwell on a field with a wide-field instrument, such that any object that passes through the field during that time – even for just a few frames – will be detected. This makes it ideal for uncued searches of the GEO belt region or for blind detection of fast-moving asteroids. When paired with the "Monitor Queue" module, it is possible to very efficiently analyze dozens of datasets captured throughout the night. More information about the underlying detection process is conveyed in the following paper:

https://storage.googleapis.com/tycho_data-1/docs/2024-sas-dparrott-paper.pdf

The "Fast Tracker" module can be operated as follows:

(1) Ensure that the dataset has been calibrated, plate solved, and aligned. Please review the earlier sections in this user guide for more information on how to perform these processes, if needed. Note that just like synthetic tracking, the Fast Tracker works best when the images have been normalized to a common background level (as is achieved through the Calibration routine).
(2) Navigate to ***Action->Fast Tracker*** from the main menu.
(3) A new window will appear, as shown in Figure 118, prompting for the desired configuration.
(4) **Motion Limits**: Specify any limits on speed and position angle here. There is typically no noticeable performance improvement by applying motion limits for the Fast Tracker. However, if you know in advance that the object(s) you wish to detect has a certain range of motion, it can sometimes be desirable to specify such a limit primarily for the purpose of filtering out other detections.

(5) **Sensitivity**: Choose between Standard and Maximum. Standard is recommended for nearly all survey operations, as the latter setting is more likely to present false detections.
(6) Click “OK” to start the Fast Tracker.

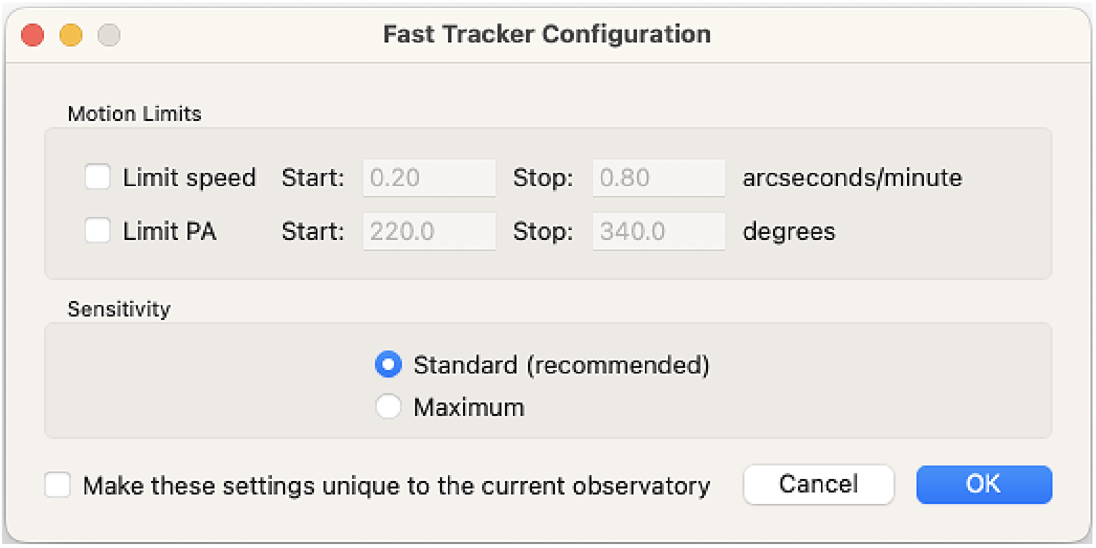
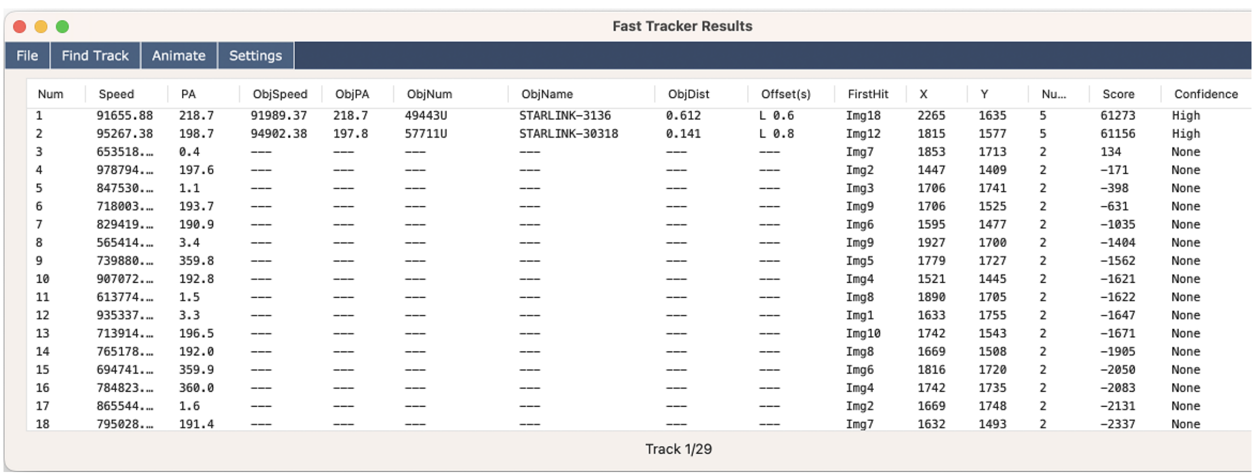


Figure 118 - Fast Tracker Configuration

After a moment (typically within a few seconds), you should see the results of the Fast Tracker operation. These results are presented in a new window labeled “Fast Tracker Results”. The list of results is sorted by confidence – detections having a higher confidence score are shown at the top, while those having a lower score (and generally more likely to be false detections) are shown at the bottom.



Num	Speed	PA	ObjSpeed	ObjPA	ObjNum	ObjName	ObjDist	Offset(s)	FirstHit	X	Y	Nu...	Score	Confidence
1	91655.88	218.7	91989.37	218.7	49443U	STARLINK-3136	0.612	L 0.6	Img18	2265	1635	5	61273	High
2	95267.38	198.7	94902.38	197.8	57711U	STARLINK-30318	0.141	L 0.8	Img12	1815	1577	5	61156	High
3	653518....	0.4	---	---	---	---	---	---	Img7	1853	1713	2	134	None
4	978794....	197.6	---	---	---	---	---	---	Img2	1447	1409	2	-171	None
5	847530....	1.1	---	---	---	---	---	---	Img3	1706	1741	2	-398	None
6	718003....	193.7	---	---	---	---	---	---	Img9	1706	1525	2	-631	None
7	829419....	190.9	---	---	---	---	---	---	Img6	1595	1477	2	-1035	None
8	565414....	3.4	---	---	---	---	---	---	Img9	1927	1700	2	-1404	None
9	739880....	359.8	---	---	---	---	---	---	Img5	1779	1727	2	-1562	None
10	907072....	192.8	---	---	---	---	---	---	Img4	1521	1445	2	-1621	None
11	613774....	1.5	---	---	---	---	---	---	Img8	1890	1705	2	-1622	None
12	935337....	3.3	---	---	---	---	---	---	Img1	1633	1755	2	-1647	None
13	713914....	196.5	---	---	---	---	---	---	Img10	1742	1543	2	-1671	None
14	765178....	192.0	---	---	---	---	---	---	Img8	1669	1508	2	-1905	None
15	694741....	359.9	---	---	---	---	---	---	Img6	1816	1720	2	-2050	None
16	784823....	360.0	---	---	---	---	---	---	Img4	1742	1735	2	-2083	None
17	865544....	1.6	---	---	---	---	---	---	Img2	1669	1748	2	-2131	None
18	795028....	191.4	---	---	---	---	---	---	Img7	1632	1493	2	-2337	None

Figure 119 - Fast Tracker Results

Identifying Objects

It is possible to invoke an identification routine on the results returned from the “Fast Tracker” module. This is achieved by navigating to ***File->Match ArtSats*** from the menu of the “Fast Tracker” window. Refer to the section labeled “Satellite Identification: Fast Tracker Results” for more details.

Fast Follow-up

During the course of a nightly survey, one may encounter an object for which additional observations are desired. One way to achieve this is to predict where the object will be located within the next minute or two – just enough time to slew the telescope to the desired coordinates, but not so much time that the object is lost. The tool to achieve this is called “Fast Follow-up”:

(1) Right-click on the object of interest in the “Fast Tracker Results” window
(2) Choose “Fast follow-up” from the dropdown menu
(3) The “Session Planner” window will appear, indicating the predicted position of the object

Note that if the object is already below 10 degrees in altitude or has entered Earth’s shadow at the time of the follow-up request, the returned message will indicate “Object not visible in next 10 minutes.”

Satellite Lookup

The satellite catalog can be conveniently searched by navigating to ***Satellites->Satellite Lookup*** from the main menu. This presents a new window as shown in Figure 120.

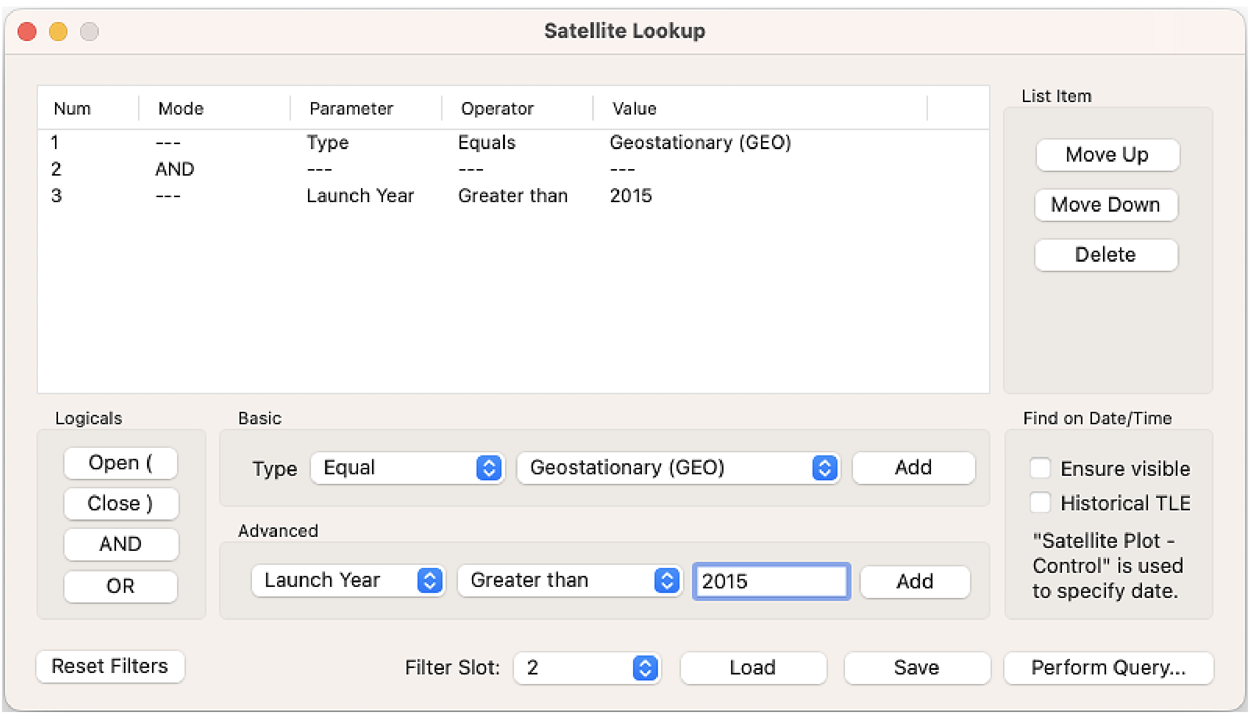
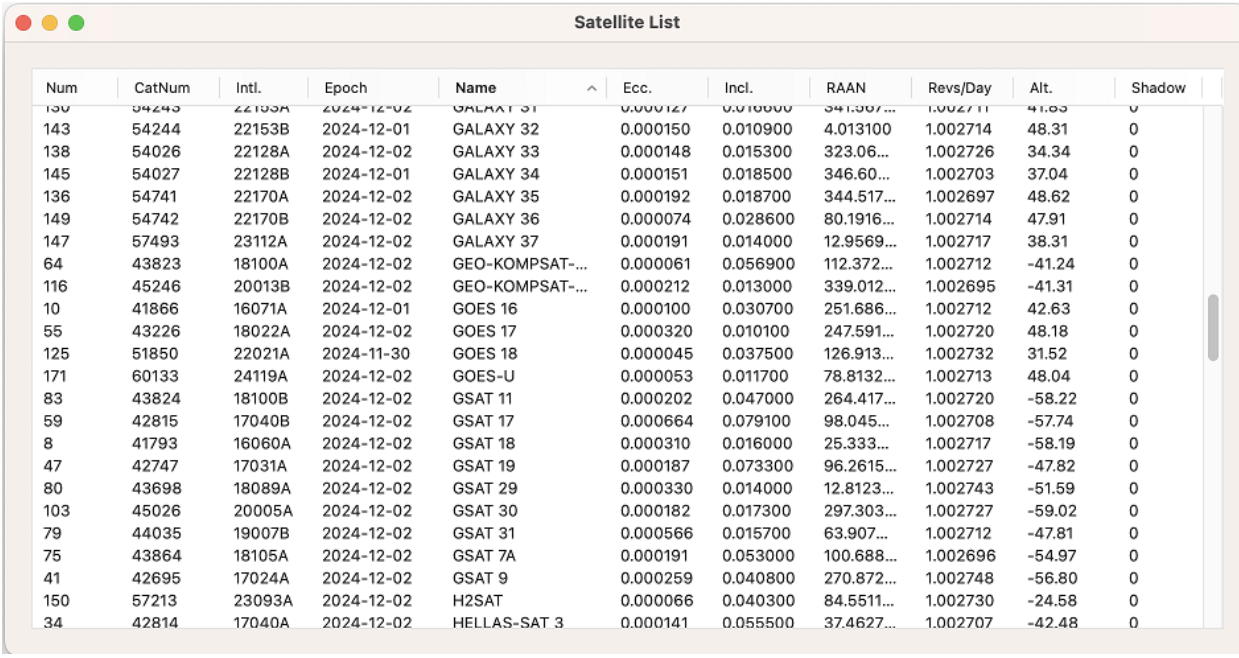


Figure 120 - Satellite Lookup

In the example shown in Figure 120, two search parameters are specified: “Type” and “Launch Year”. This shows a query requesting all Geostationary satellites that launched after the year 2015. Additional parameters are added by first specifying a logical (AND/OR, as well as any surrounding parentheses as needed), followed by the parameter (basic or advanced). When finished, click “Perform Query” to submit the lookup request. After a moment the results will be returned as shown in Figure 121. Note that there is a limit of 1000 results displayed.



Satellite List

Num	CatNum	Intl.	Epoch	Name	Ecc.	Incl.	RAAN	Revs/Day	Alt.	Shadow
143	54244	22153B	2024-12-01	GALAXY 32	0.000150	0.010900	4.013100	1.002714	48.31	0
138	54026	22128A	2024-12-02	GALAXY 33	0.000148	0.015300	323.06...	1.002726	34.34	0
145	54027	22128B	2024-12-01	GALAXY 34	0.000151	0.018500	346.60...	1.002703	37.04	0
136	54741	22170A	2024-12-02	GALAXY 35	0.000192	0.018700	344.517...	1.002697	48.62	0
149	54742	22170B	2024-12-02	GALAXY 36	0.000074	0.028600	80.1916...	1.002714	47.91	0
147	57493	23112A	2024-12-02	GALAXY 37	0.000191	0.014000	12.9569...	1.002717	38.31	0
64	43823	18100A	2024-12-02	GEO-KOMPSAT-...	0.000061	0.056900	112.372...	1.002712	-41.24	0
116	45246	20013B	2024-12-02	GEO-KOMPSAT-...	0.000212	0.013000	339.012...	1.002695	-41.31	0
10	41866	16071A	2024-12-01	GOES 16	0.000100	0.030700	251.686...	1.002712	42.63	0
55	43226	18022A	2024-12-02	GOES 17	0.000320	0.010100	247.591...	1.002720	48.18	0
125	51850	22021A	2024-11-30	GOES 18	0.000045	0.037500	126.913...	1.002732	31.52	0
171	60133	24119A	2024-12-02	GOES-U	0.000053	0.011700	78.8132...	1.002713	48.04	0
83	43824	18100B	2024-12-02	GSAT 11	0.000202	0.047000	264.417...	1.002720	-58.22	0
59	42815	17040B	2024-12-02	GSAT 17	0.000664	0.079100	98.045...	1.002708	-57.74	0
8	41793	16060A	2024-12-02	GSAT 18	0.000310	0.016000	25.333...	1.002717	-58.19	0
47	42747	17031A	2024-12-02	GSAT 19	0.000187	0.073300	96.2615...	1.002727	-47.82	0
80	43698	18089A	2024-12-02	GSAT 29	0.000330	0.014000	12.8123...	1.002743	-51.59	0
103	45026	20005A	2024-12-02	GSAT 30	0.000182	0.017300	297.303...	1.002727	-59.02	0
79	44035	19007B	2024-12-02	GSAT 31	0.000566	0.015700	63.907...	1.002712	-47.81	0
75	43864	18105A	2024-12-02	GSAT 7A	0.000191	0.053000	100.688...	1.002696	-54.97	0
41	42695	17024A	2024-12-02	GSAT 9	0.000259	0.040800	270.872...	1.002748	-56.80	0
150	57213	23093A	2024-12-02	H2SAT	0.000066	0.040300	84.5511...	1.002730	-24.58	0
34	42814	17040A	2024-12-02	HELLAS-SAT 3	0.000141	0.055500	37.4627...	1.002707	-42.48	0

Figure 121 - Satellite Lookup Results

It is also possible to right-click on a satellite in the returned list and choose an option from the popup menu that appears:

Add to TLE Propagator: This option adds the selected satellite to the TLE Propagator module. This can be useful if it is desirable to work with the satellite on a regular basis.
Attach to Dataset: This option computes the predicted position and velocity of the satellite using its orbital elements and attaches that information to the current dataset. Convenience features such as automatically centering the satellite in the image become available with this routine.
Attach to Session Planner: This option displays the Session Planner module with the table of entries corresponding to the ephemeris of the selected satellite.
Attach to Satellite Viewer: This option displays the Satellite Viewer module for rendering the orbit of the selected satellite.

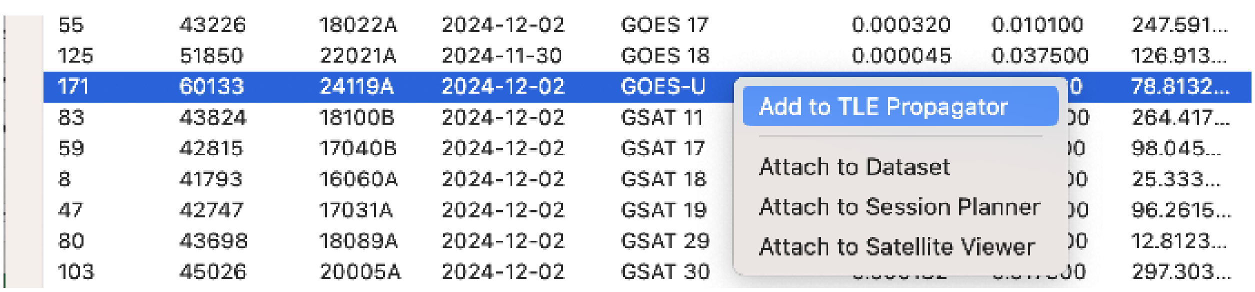


Figure 122 - Available Actions on Satellite Lookup Results

Note that the three options "Attach to…" are also available in the TLE Propagator module. So if the satellite is added to the TLE Propagator module, these functions can be invoked there as well.

TLE Propagator

The TLE Propagator module acts as a repository to store and maintain elements of artificial satellites for convenient access. Satellites can be added in two ways:

(1) Via the “Satellite Lookup” tool. Right-click on a returned satellite in the results and choose “Add to TLE Propagator”.
(2) Manually inputting a two-line element (TLE). A TLE can be input into the edit box near the bottom of the TLE Propagator window, and then click the button labeled “Add New TLE to List”.

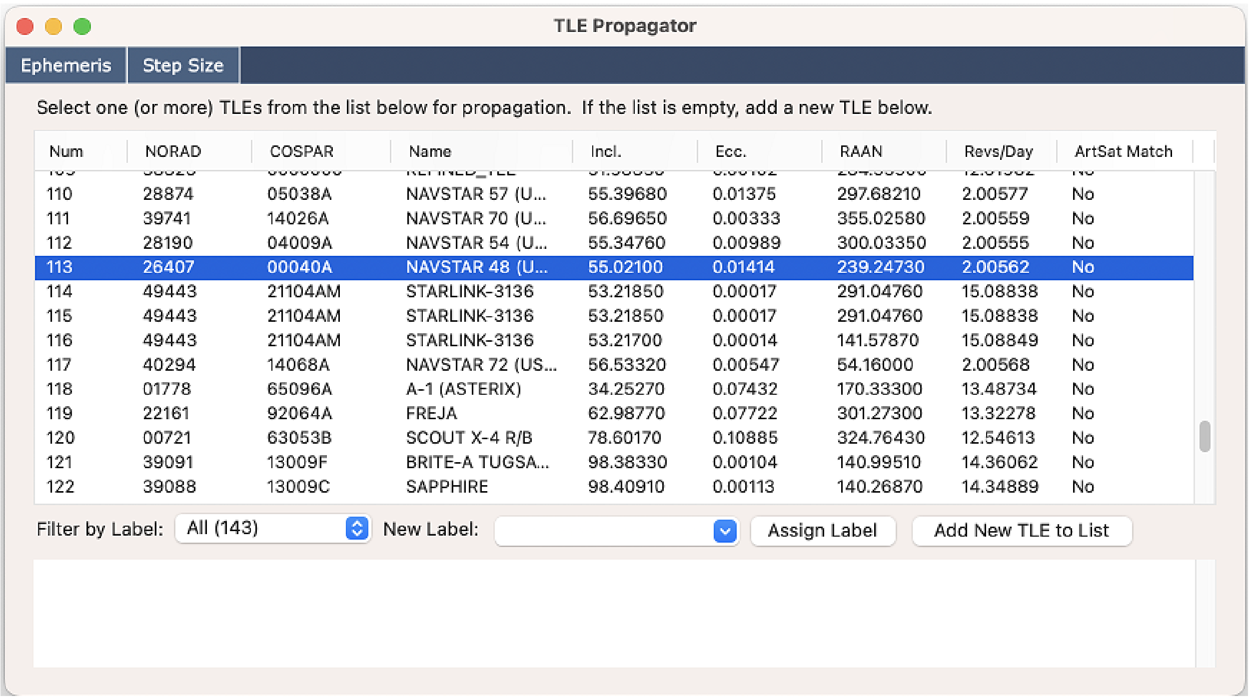


Figure 123 - TLE Propagator

There are a number of actions that can be performed on satellites added to the TLE Propagator, as shown in Figure 124.

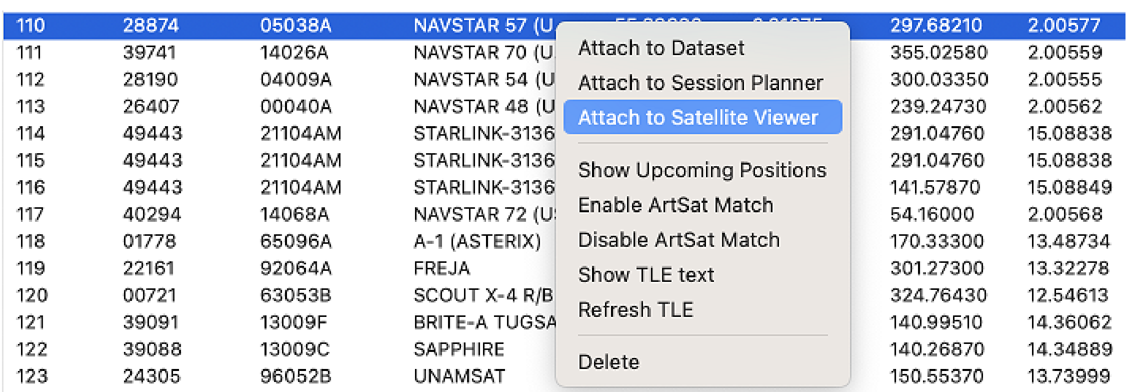


Figure 124 - Available Actions for TLE Propagator Items

Attach to Dataset: This option computes the predicted position and velocity of the satellite using its orbital elements and attaches that information to the current dataset. Convenience features such as automatically centering the satellite in the image become available with this routine.

Attach to Session Planner: This option displays the Session Planner module with the table of entries corresponding to the ephemeris of the selected satellite.
Attach to Satellite Viewer: This option displays the Satellite Viewer module for rendering the orbit of the selected satellite.
Show Upcoming Positions: This option is similar to "Attach to Session Planner", except it also automatically sets the start and stop times to be Now+24 hours, with a 1-minute interval. It is a convenience function for those wanting to know the positions of the satellite over the next 24 hours.
Enable ArtSat Match: This option includes the selected satellite in the list of satellites to be searched, provided that the "ArtSat Match Configuration" setting enables the option to match TLEs from the TLE Propagator.
Disable ArtSat Match: This option excludes the selected satellite from the list of satellites to be searched (default setting).
Show TLE text: This option updates the edit box with the TLE of the selected satellite.
Refresh TLE: This option will check to see if a newer TLE is available for the selected satellite.
Delete: Removes the selected satellite from the list.

Satellites (more specifically "TLEs" or "elsets") can also be assigned a label to assist with organizing the list of items shown in the TLE Propagator. For example, you might be working on constructing a new orbit of a recently detected object, for which you might generate multiple TLEs: an initial orbit derived from the first set of observations, and a refined orbit derived from additional observations. To assign a label to one or more TLEs:

(1) Select the TLE(s) in the list for which a new label is to be assigned
(2) Specify the desired label in the "New Label" edit box
(3) Click the button "Assign Label"

Once a label has been assigned, the list can then be filtered by that label by choosing the appropriate label from the "Filter by Label" dropdown. To remove a label on a selected item, specify a blank label in the "New Label" edit box and click "Assign Label". A blank label removes any existing assigned label on an item.

Satellite Viewer

The Satellite Viewer module offers multiple ways to view the orbit of one or more satellites. To access this module, navigate to ***Satellites->Satellite Viewer*** from the main menu. It can also be accessed whenever the "Attach to Satellite Viewer" menu open is chosen from the popup menu on a selected satellite. The first time that the Satellite Viewer is launched, it will download a map image file used for the "World Map" rendering option. This may take a minute or two to download. Once completed, it is rather quick to switch between the various view options:

(1) World Map
(2) RA/Dec
(3) Alt/Az
(4) Top View (3D)
(5) Side View (3D)

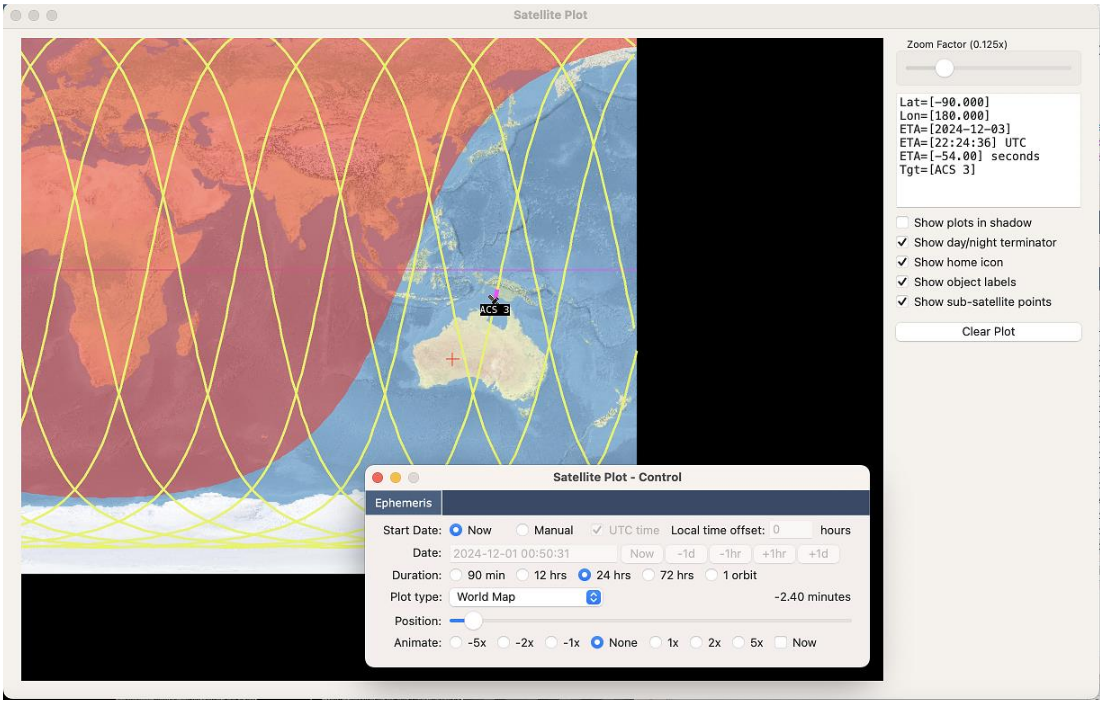


Figure 125 - World Map

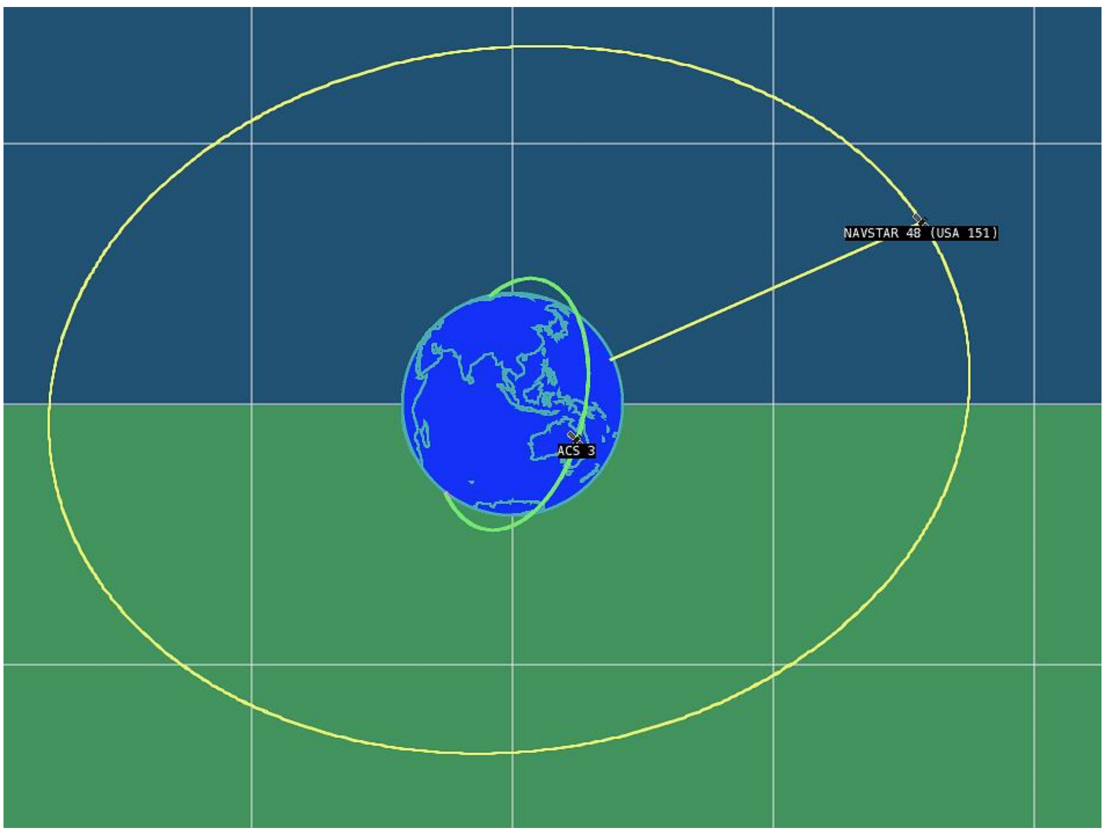


Figure 126 - Top View, Showing Two Satellites (ACS3 and NavStar 48)

The current view can be changed by right-clicking in the view and choosing the desired view from the popup menu that appears.

The “Satellite Plot – Control” window provides a way to specify the start time of the plot as well as its duration (90 minutes, 12 hours, 24 hours, 72 hours, or 1 orbit). It is also possible to advance the satellite position through the slider control, or by choosing one of the animation radio buttons to adjust the animation rate, with the “Now” checkbox updating the rate to what it would be in real-time.

Another common use case of the “Satellite Plot – Control” window is to determine which satellites are currently visible, or will be visible at a specified time. For example, set the “Start Date” to “Manual”, then click the “Now” button next to the edit box in the next row. This sets the manual date to the current time. Then click the “+1hr” button to advance it to one hour in the future (or simply type in the desired date/time manually). Once the date/time has been specified, navigate to ***Satellites->Satellite Lookup*** from the main menu and specify the desired search criteria, such as “Low Earth (LEO)” for Type. Check the box “Ensure Visible” and then click “Perform Query”. You will now be presented with a list of LEO satellites that are visible at your location at the specified date/time.

IOD Report (Observations)

The IOD Report module provides the key functions to operate on a set of observations. Access the IOD Report module by navigating to ***Satellites->Observations*** from the main menu. IOD stands for “Interactive Orbit Determination” and was developed in 1998 by George Lewis. It remains the most common observation format among satellite watchers. Consequently, the IOD Report module also provides several conversion routines to transform observations in other formats into IOD format:

(1) MPC1992
(2) ADES
(3) RDE
(4) UK

These conversion routines are accessed by navigating to the appropriate menu item under the ***Convert*** menu from the “IOD Report” window.

GNSS Analysis

One of the features of the IOD Report module is to evaluate a set of observations on a Global Navigation Satellite System (GNSS) object. There are presently three supported GNSS constellations: NavStar, GLONASS, and Galileo. By taking measurements of a known GNSS satellite, one can determine the cross-track and along-track errors, which can help assess the accuracy of the observations in both position and timing.

(1) Select a GNSS satellite that is visible from your location.
(2) Generate measurements (observations) of this satellite.
(3) Supply the resulting observations into the IOD Report.
(4) Choose ***GNSS Analysis->Evaluate Observations*** from the menu of the “IOD Report” window.

The first step – selecting a GNSS satellite – can be done by making use of the “Satellite Lookup” tool. Navigate to ***Satellites->Satellite Lookup*** from the main menu. From the window that appears, clear out any existing entries in the lookup parameters. Next, in the “Basic” group, select “Equal” for the Type comparison dropdown and choose "GNSS (Astrometry)" for the Type selection. Then click the "Add" button to add the lookup parameter. Next, check the box labeled "Ensure Visible" for the "Find on Date/Time" group, and make sure to uncheck "Historical TLE" (if it is checked). Finally, click "Perform Query" to submit the lookup request. After a few seconds the resulting GNSS satellites that are supported by the astrometry analysis are returned – specifically, those that are visible at the date/time specified on the "Satellite Plot – Control" window. If you need to adjust the date/time for which you intend to view the satellite, make the adjustment in the "Satellite Plot – Control" window and then re-submit the lookup request by clicking the "Perform Query" button once again.

Using the provided list of satellites in the "Satellite List" window, choose a satellite that will be at a reasonable altitude above the horizon (you can click the "Alt." column to sort the list by altitude). Right-click on the desired satellite and choose "Add to TLE Propagator". By adding it to the TLE Propagator, you can keep the satellite elements stored for easy access later on. Now, at the TLE Propagator window, right-click on the added satellite and choose "Attach to Session Planner". You can then adjust the "Start Date" and "Stop Date" on the "Session Planner" window and then choose "Attach to Session Planner" once more with the updated dates as needed. At this point, the "Session Planner" should show a list of satellite positions at the displayed times for the active observatory.

Once you have generated observations of the satellite, supply them to the "IOD Report" and choose ***GNSS Analysis->Evaluate Observations*** from the menu of the "IOD Report" window.

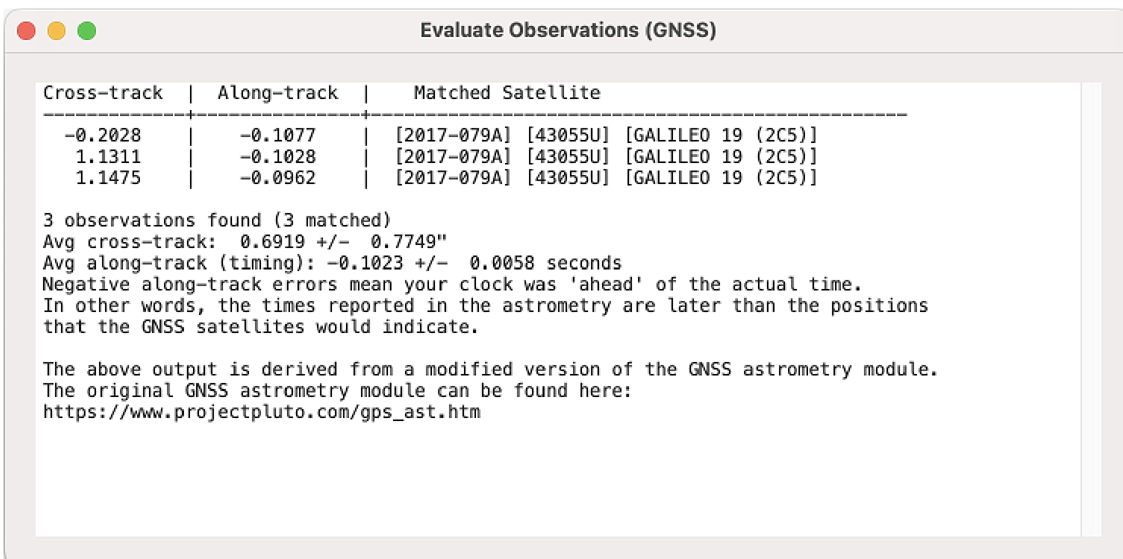


Figure 127 - GNSS Astrometry Results

Inspiration for the GNSS Analysis module came from the "Check GNSS astrometry" tool, available here: https://www.projectpluto.com/gps_ast.htm

One improvement that has been made is that the GNSS Analysis module in Tycho supports observations in the IOD format.

TLE Analysis

The "TLE Analysis" component of the "IOD Report" offers the following features:

(1) Generate Initial TLE
(2) Refine Existing TLE
(3) Evaluate Observations
(4) Find Matching ArtSat

All of these features operate on the currently supplied observations in the "IOD Report". For example, the "Find Matching ArtSat" feature, which has been discussed previously in the section labeled "Satellite Identification: IOD Report (Observations)", will attempt to identify the satellite associated with the supplied observations.

Generate Initial TLE

When an object is initially observed, and there is no known orbit yet established, one can proceed to perform initial orbit determination using either two or three observations. Two observations can be acceptable if one suspects that the object has a circular orbit. However, three observations are preferable if the object has notable eccentricity, and can tend to produce a more robust initial orbit on the condition that the observations are spaced apart at reasonable intervals. If the observations are too close together in time (that is, a short arc compared to the orbit of the object), then the three observations will likely produce an invalid initial orbit.

As an example, consider an object having approximately 2 revolutions per day (revs/day). In this scenario it can be preferable to space the three observations apart by at least 20-30 minutes to provide a usable sampling of the orbit of the object. By contrast, an object having 15 revs/day might only need the observations to span a few minutes to achieve a reasonable sampling of the orbit, due to its faster orbital period.

The initial TLE can be generated by supplying the observations into the "IOD Report" and then navigating to ***TLE Analysis-> Initial TLE - Single Range*** from the menu of the "IOD Report" window. If the solution failed to converge an error will be presented. Otherwise, a new TLE will be added to the "TLE Propagator" module with the label "USER_TLE". Note that just because no error is presented, it does not mean that the initial TLE is accurate; the accuracy is again dependent on the quality of the observations and the arc of the orbit that has been sampled.

Tycho v13 introduces an additional method to generating an initial TLE, known as "Range-Swept Differential Correction" (RSDC). This algorithm is more robust than the conventional IOD method as it evaluates hundreds of candidate orbits, each exploring a different range value. It is particularly ideal when the object has a non-circular orbit. More information about the RSDC method can be found here: https://downloads.tycho-files.com/docs/2025-06-29_Paper_dparrott_RSDC.pdf

To generate a TLE using the RSDC method, navigate to ***TLE Analysis->Initial TLE - Multi Range*** from the menu of the "IOD Report" window. This method works well when one has collected numerous measurements of an object from the same night (a dozen or more measurements, typically).

Refine Existing TLE

After an initial TLE has been generated, the next step is to gather additional observations to refine the computed orbit in a process known as Differential Correction. Differential Correction can be thought of as essentially a multi-dimensional Newton-Raphson routine that endeavors to find the optimal adjustments to be made to the elements of an existing TLE that would allow it to fit to a full set of observations. Note that it does rely on having a reasonably good estimate for the initial TLE – in other words, one cannot expect to start with a random set of elements and reasonably expect it to converge to the desired elements. In practice this generally amounts to a process similar to the following:

(1) Initial orbit determination using observations at first detection (either using ***TLE Analysis->Initial TLE – Single Range*** or ***TLE Analysis->Initial TLE – Multi Range***)
(2) Collect additional observations on the object
(3) Differential correction using the combined set of observations (***TLE Analysis->Refine TLE - Automatic***)

Navigate to ***TLE Analysis->Refine TLE - Automatic*** from the menu of the "IOD Report" window. Specify the TLE to be refined in the "Existing TLE" edit box. Then, after making sure that the observations you want are supplied to the IOD Report, click the "Start" button to proceed. It should (ideally) arrive at a refined orbit determination using the supplied observations and the TLE provided. You can also click the button "Copy Output to TLE Propagator" to add the refined TLE to the TLE Propagator module.

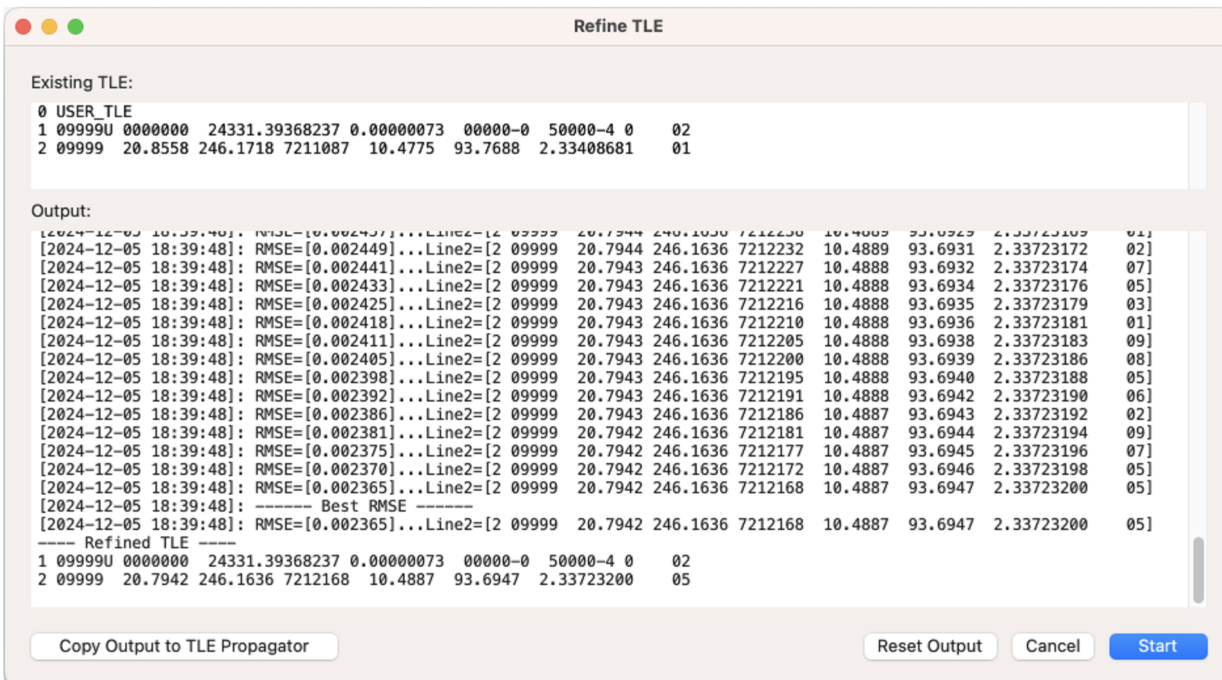


Figure 128 - TLE Refinement using Differential Correction

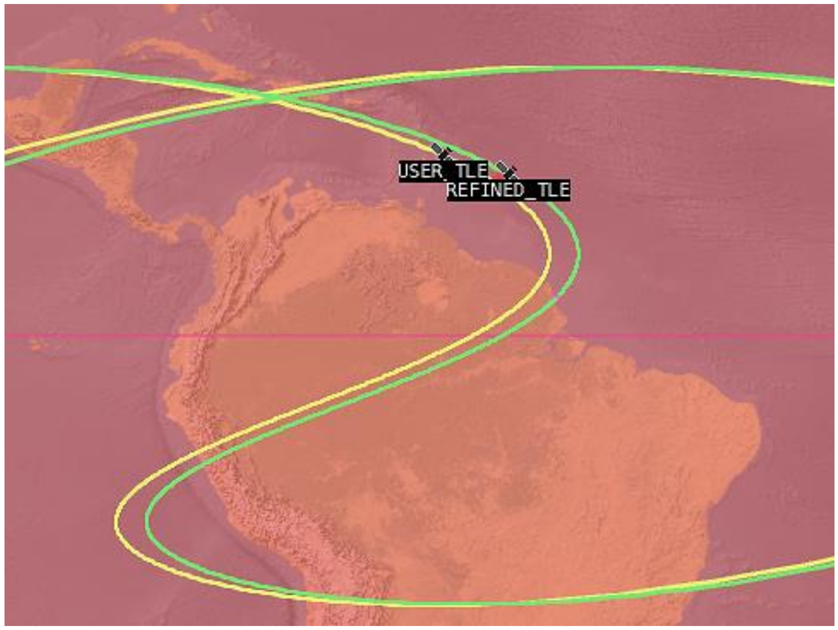


Figure 129 - Visual Comparison: USER_TLE (Initial Orbit) and REFINED_TLE (Refined TLE)

Figure 129 shows a visual comparison of the initial orbit and the refined orbit. This particular object – that of a rocket body orbiting at 2.33 revolutions per day – is in a rather eccentric orbit at e=0.72. Using the initial orbit, it was possible to recover it a few days later, albeit with rather large uncertainty on the time of arrival: the object made an appearance near the expected coordinates approximately 25 minutes earlier than predicted. However, after refining the orbit, its recovery on subsequent nights was much easier, as it arrived "on time" – within seconds – relative to the predicted time of arrival.

Troubleshooting

Windows operating system: If you find that a window has an incorrect size, restart Tycho and navigate to ***Window->Reset Window Positions*** from the main menu. Be sure to do this on a fresh launch of Tycho, prior to opening any new windows. This will reset any previously saved window position information.

Additional Resources

This guide has covered the basics of how to use the Tycho software. There are also tutorial videos on the website at https://www.tycho-tracker.com

Finally, if you run into any issues or have questions or comments, please feel free to contact me using the "Contact" page on the website, or try out the new forum:

https://groups.google.com/forum/#!forum/tycho-tracker