User Guide for Tycho v10.2

By Daniel Parrott

Last updated February 21, 2023

Contents

Introduction

Welcome to Tycho. This software is designed to facilitate the detection and measurement of asteroids, comets, and variable stars. It also supports a technique known as Synthetic Tracking, which enables the detection of very faint 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. This is made possible by using the graphics processing unit (GPU) on your computer.

Starting with v5.3, it is now possible to run the software without a dedicated graphics card. However, it is still recommended to equip your machine with a dedicated graphics card if you intend to perform synthetic tracking. Most other functionality can be done acceptably without GPU acceleration.

System Requirements

- CPU: Intel Core i5 4590 or better
- RAM: 8GB or more
- Windows 7 SP1 or better

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 much 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 on both AMD and NVIDIA graphics cards.

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 a dedicated video card (GPU), such as NVIDIA or AMD, then it is recommended to enable GPU acceleration. This is accomplished by going to *OpenCL->Device Selector* from the main menu.

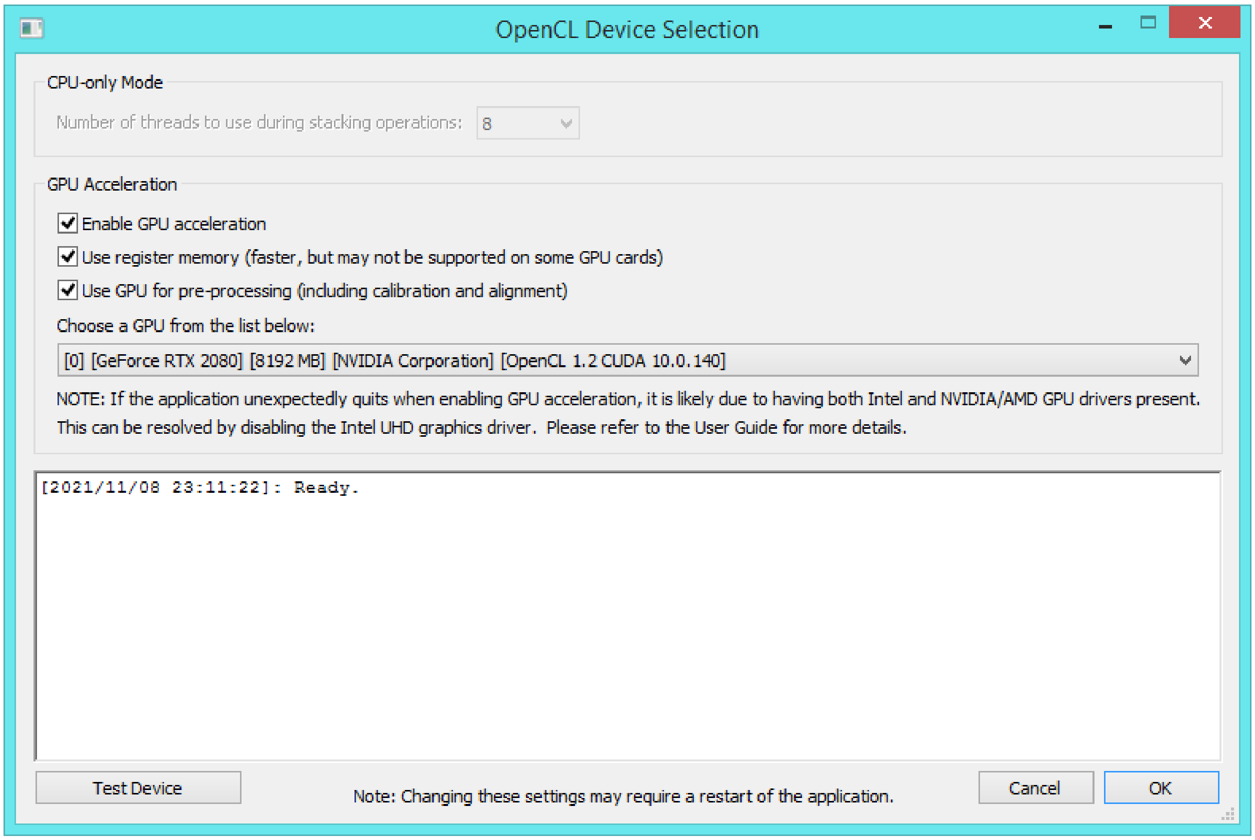


Figure 1 - OpenCL Device Selector

Troubleshooting

1) If you get an error message when choosing a GPU device, make sure you have the latest drivers installed for your 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 your 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:

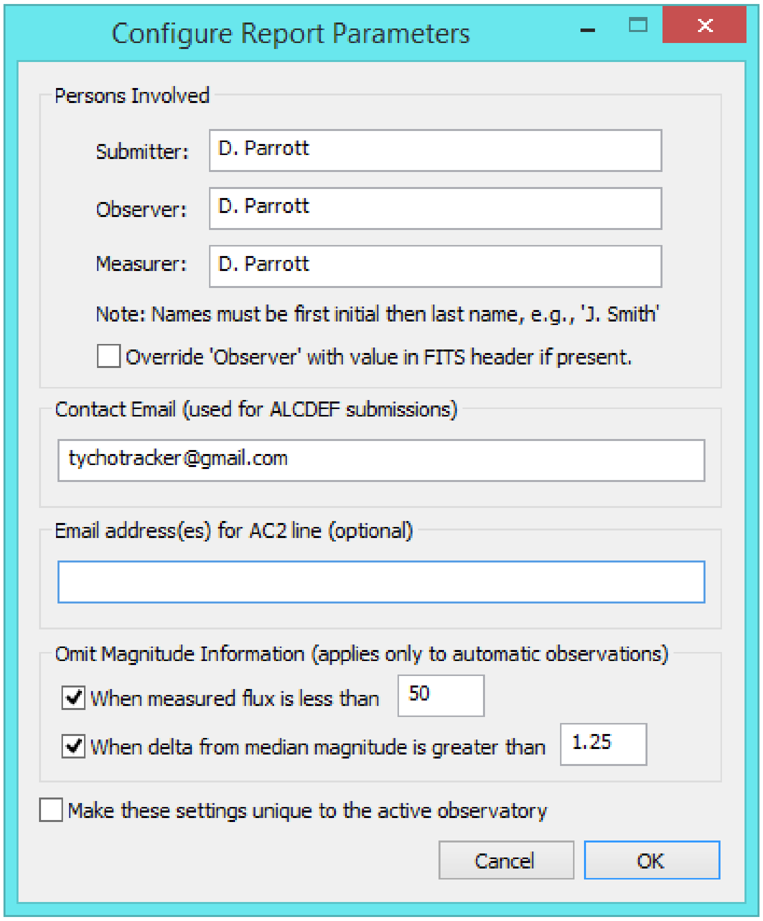


Figure 2 - Report Parameters

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.

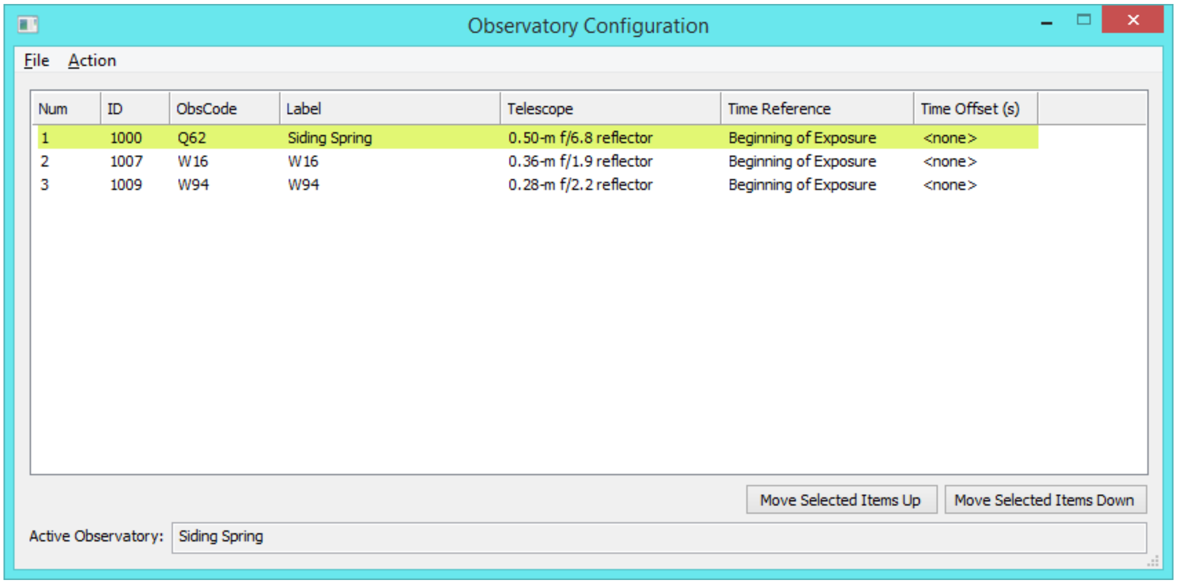


Figure 3 - Observatory Configuration

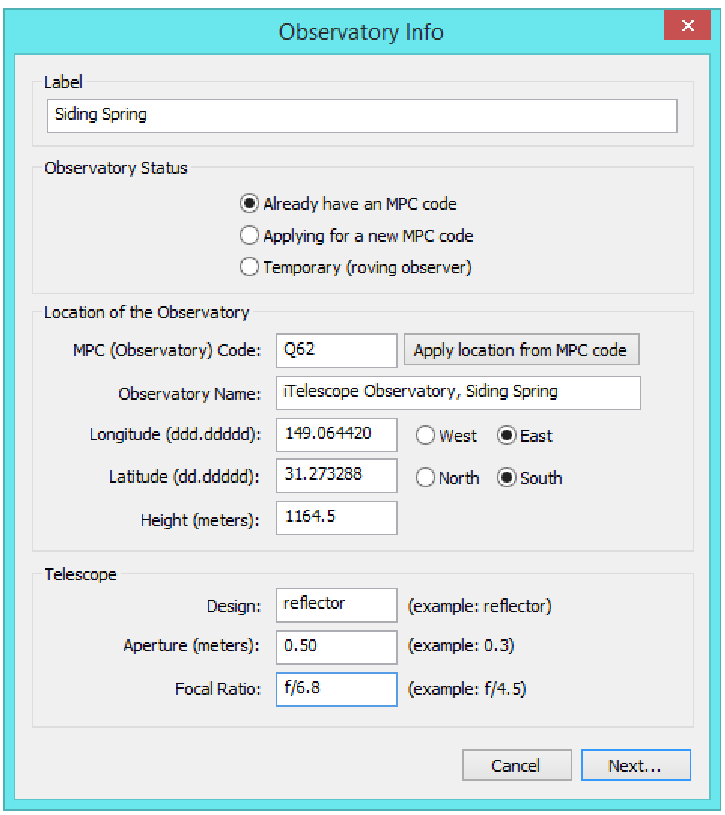


Figure 4 - Add New Observatory

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.

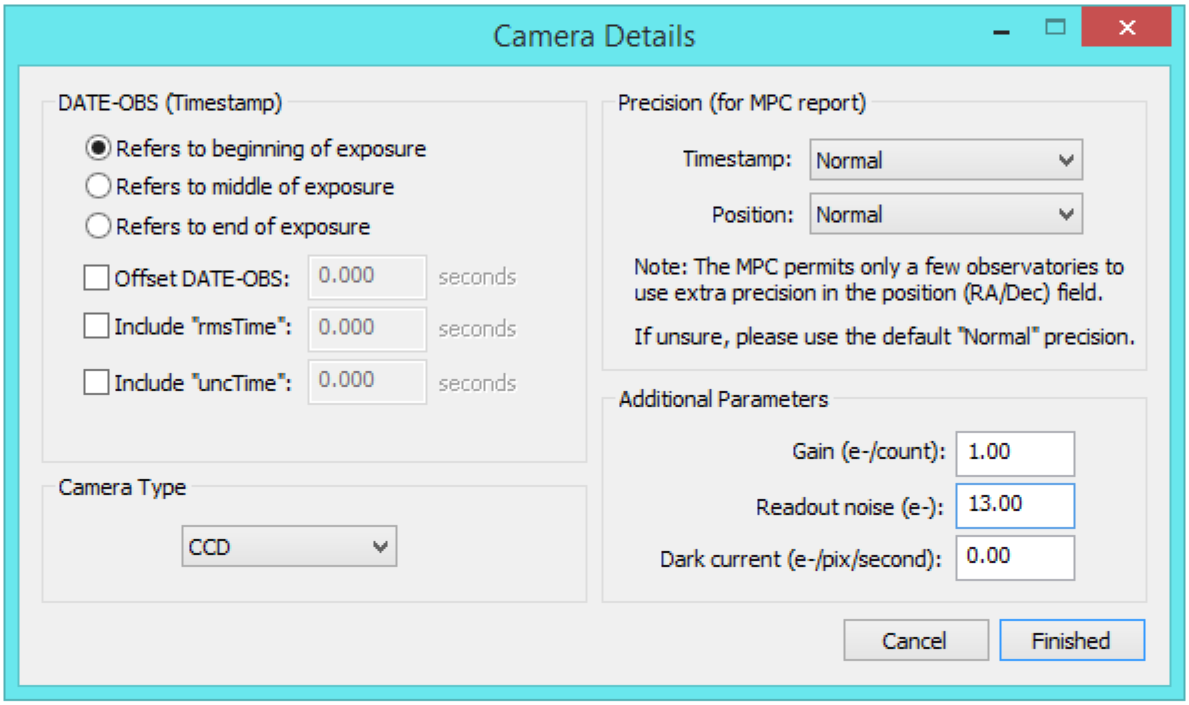


Figure 5 - Camera Details

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.

CCD Characteristics: These parameters are used for SNR calculation.

Finally, click “OK” 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 primary purpose of Tycho is to detect and measure asteroids. After you have generated measurements, it is generally desirable to compare them with other measurements for validation. The "*Find_Orb*" software, written by Bill Gray, is quite useful in this regard.

If you have not already done so, please download the **modified** version of the *Find_Orb* software from the Tycho website:

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

Also, **do not** place *Find_Orb* in the root directory (C:\), as that would require elevated privileges. Instead, place it in a folder such as the Desktop, e.g., C:\Users\Daniel\Desktop\find_o64\.

Once you have downloaded *Find_Orb*, navigate to *Settings->Find_Orb* and configure the file path similar to how it is presented in Figure 6:

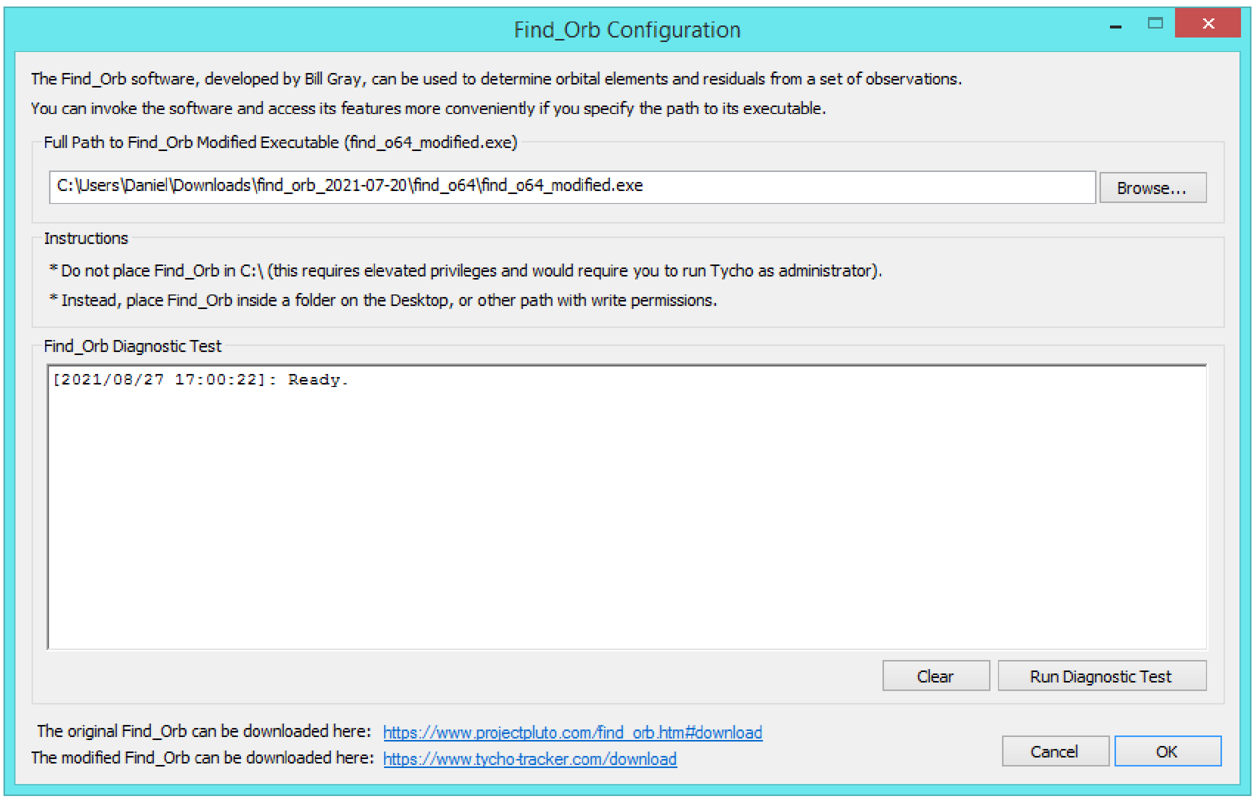


Figure 6 - Configuring the Find_Orb Software

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.

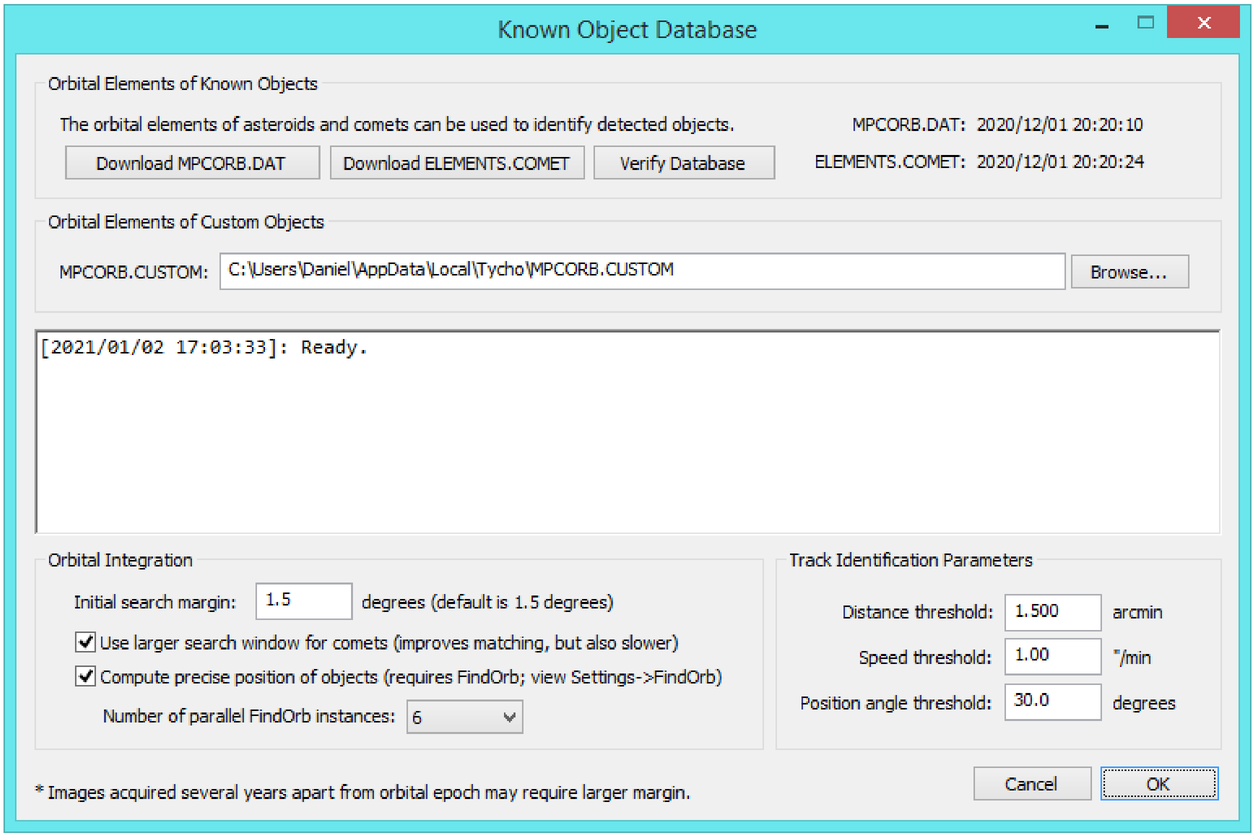


Figure 7 - Known Object Database Configuration

In the top-left there is a button "Download MPCORB.DAT". Click this button to download the orbital elements of over 1 million asteroids (approximately 60MByte in size). When it has completed, proceed to click the button "Download ELEMENTS.COMET". Finally, when that has completed, click "Verify Database". There should be over 1 million asteroids and over 900 comets listed.

There is also a path to a file labeled "MPCORB.CUSTOM". This 'custom' variant of MPCORB allows you to put in the orbital elements of objects that are not listed in the official MPCORB file, but will still be recognized as a valid MPCORB file for the purpose of object matching. You can add custom elements either by directly modifying this file, or by going to *Tools->Existing Observations* and choosing *Orbital Elements->Add to MPCORB.CUSTOM*. Note that this requires *Find_Orb* to be configured.
Set the other settings as shown in Figure 7.

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.

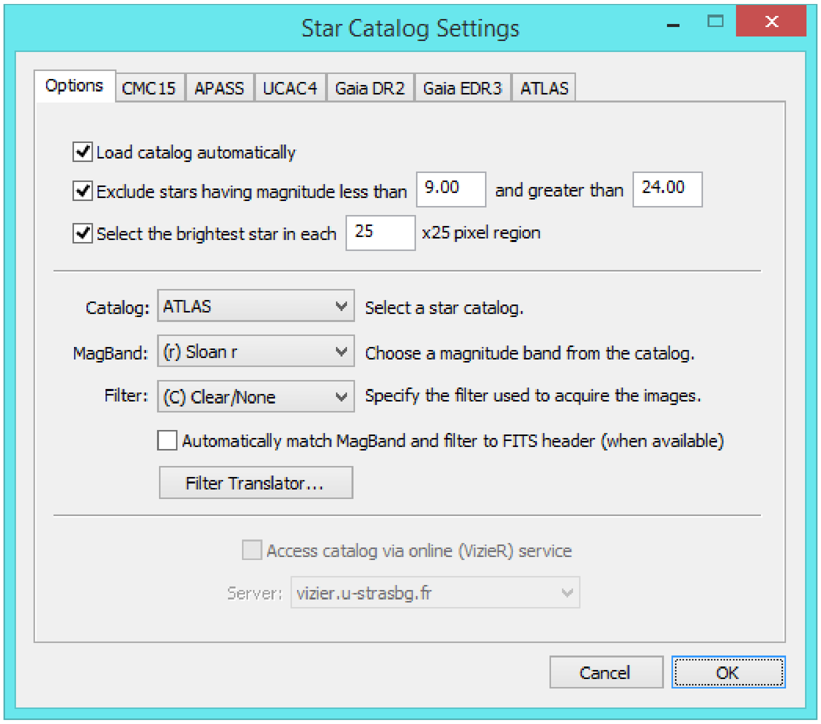


Figure 8 - Configuring the Star Catalog

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 Software

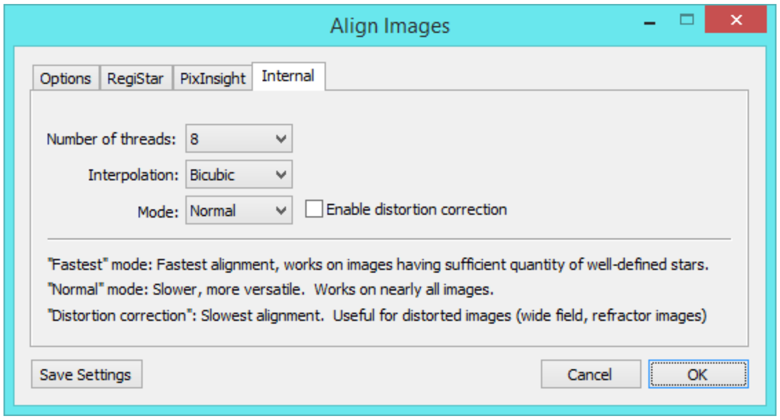


Figure 9 - Alignment Settings

Navigate to *Action->Align Images* to specify the settings for the alignment routine. You can choose to use an external program such as *RegiStar* or *PixInsight*, or you can use the internal alignment routine. If using an external program, navigate to its tab and specify the path to its executable. If using the internal routine, navigate to the "Internal" tab and select the desired options. Click "Save Settings" to continue.

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 button labeled "Download ObsCodes.html". Then, if you have configured *Find_Orb*, you can also keep it updated by clicking the button labeled "Copy to FindOrb Directory". The button labeled "Verify Path" allows you to validate that the file was downloaded correctly. As of January 2, 2021, there were 2248 observatory codes specified in the file.

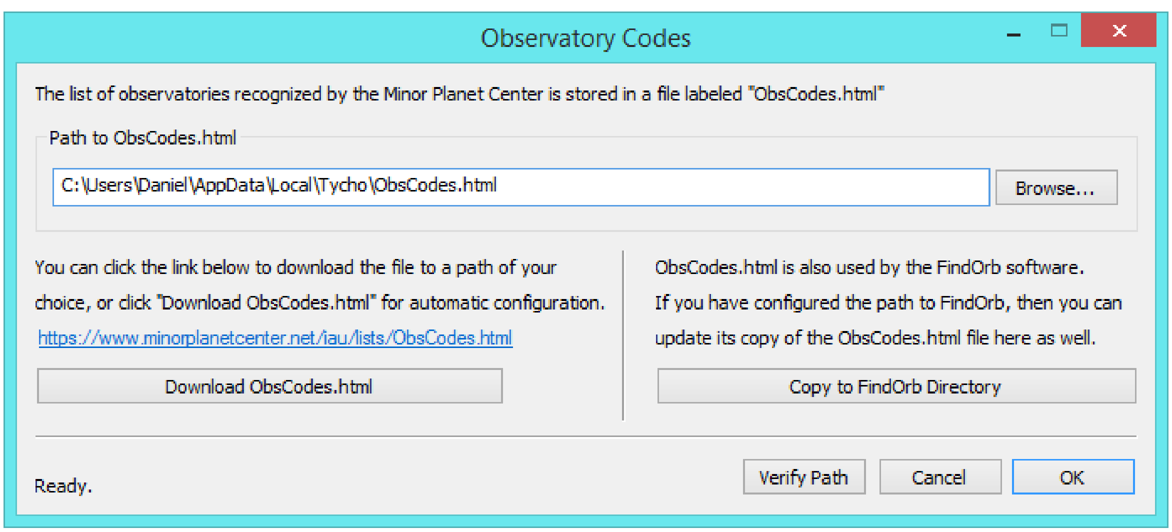


Figure 10 - Loading Observatory Codes

Configure the Plate Solver

Plate solving is necessary an order to determine image characteristics such as plate scale, orientation, and sky coordinates. Since plate solving is performed after alignment, it is only necessary to plate solve the first image in the sequence.

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 three 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 download the solve-field software onto your own machine, along with the necessary index files. This option may be useful for those who do not have a reliable Internet connection. Finally, the third option is to use the astrometry.net online service, being sure to specify "tweak_order=3" for the 3rd-order polynomial. This option is the slowest as it requires uploading the full image data.

Plate Solve Option 1: Using the Tycho Online Service

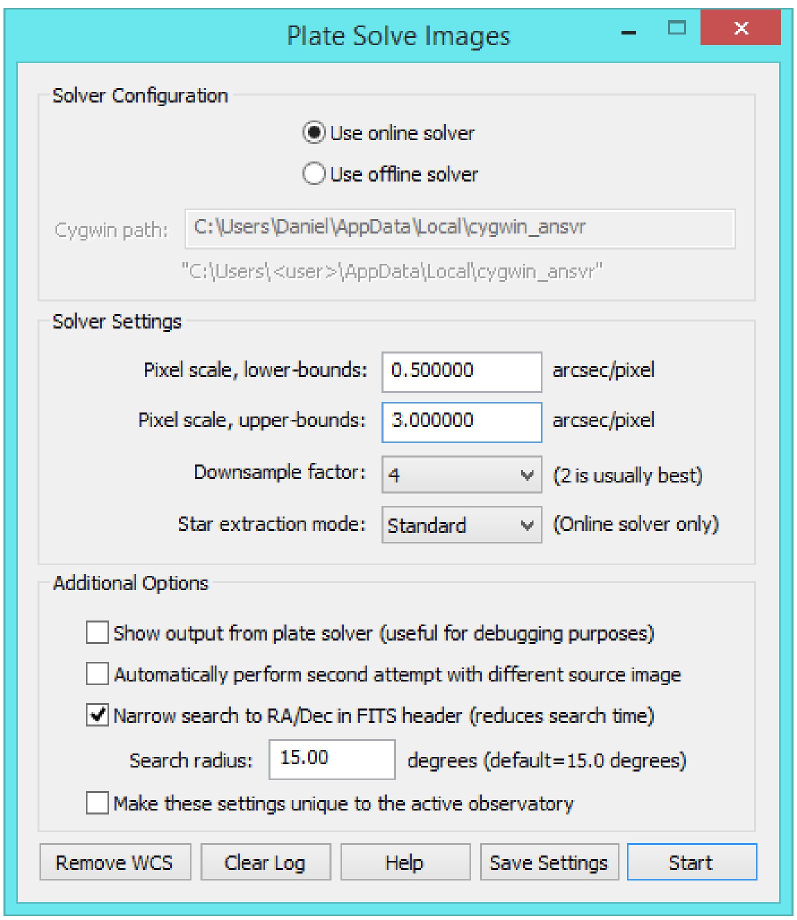


Figure 11 - Settings for Tycho Online Solver

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.

Note:
The default Downsample factor is 2, and usually works. However, if you have large images (3000x3000 or larger) then it may be useful to try a Downsample factor of 4. Conversely, if you have small images (700x700 or smaller) then it may be useful to try a Downsample factor of 1.

Also new in v9.2 is the ability to select "Standard" or "Extended" mode for star extraction. This applies only to the online solver. If you have smaller images, or are experiencing issues with getting a solution returned, you may wish to try the "Extended" extraction mode.

Plate Solve Option 2: Using the Offline Plate Solver

The offline solver may be desired by users who do not have a reliable Internet connection. For this option, the first step is to download the offline solver, a mirror of which is available from the Tycho website via the Download page from: https://www.tycho-tracker.com/download

Click on the "[ZIP]" link next to the offline solver entry, and you will start downloading the software for the offline solver. Once downloaded, extract the archive and run the setup executable. When the setup has completed, be sure to check the box "Run the Astrometric Index Downloader".

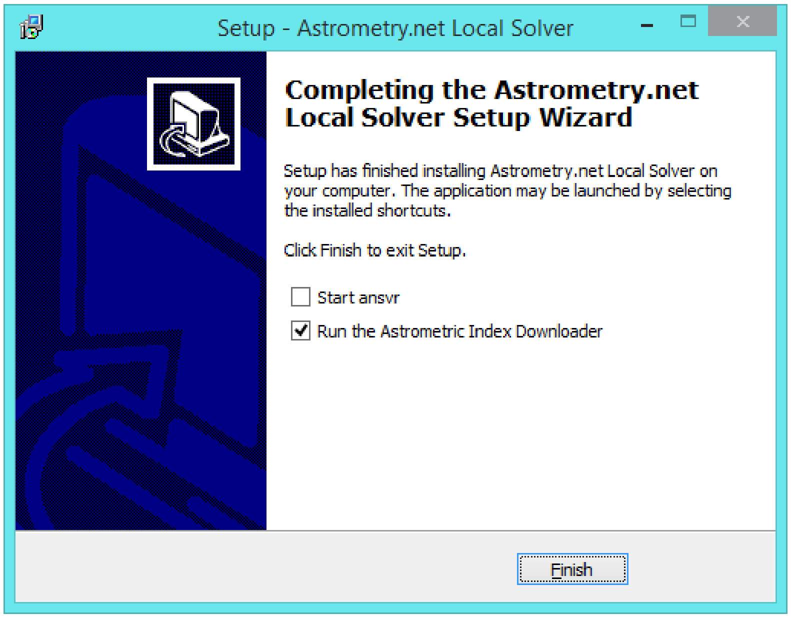


Figure 12 - Installing the Offline Plate Solver

It is not necessary to "Start ansvr" at this time, so you can uncheck that box.

The Index Downloader is important because it will automatically download the index files required for your field of view. Index files are used by the solver to perform the plate solve routine, and without them the solver will not function.

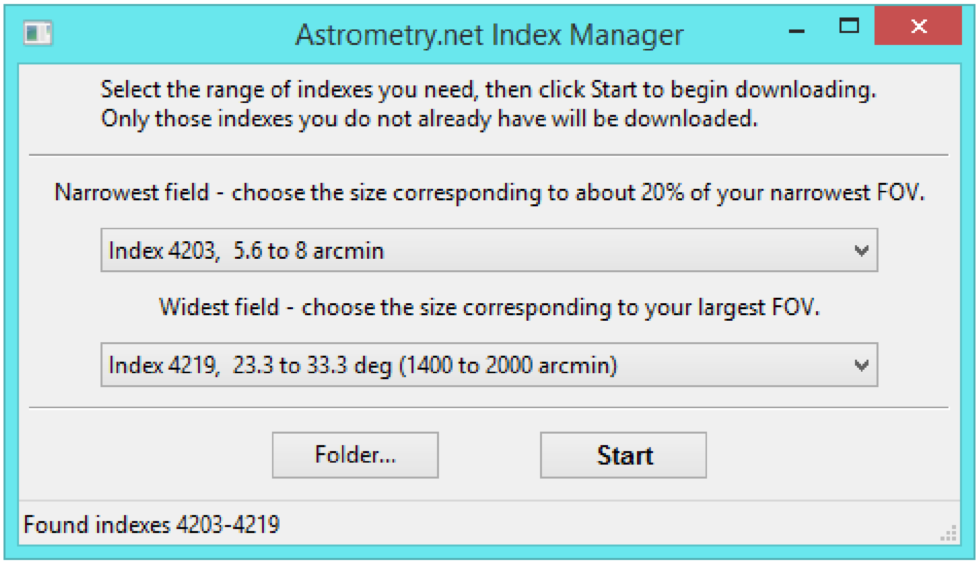


Figure 13 - Index Manager

Click "Start" once you are satisfied with the settings for "Narrowest field" and "Widest field". The Index Downloader will then proceed to download the relevant index files. When it is finished, you can close the window.

It is also recommended to install the 4100 series of index files. They are available here:

- http://broiler.astrometry.net/~dstn/4100/
- There are 13 .fits files totaling 340MB in size
- After downloading the files, place them into your index file directory (this will be similar to C:\Users\<user>\AppData\Local\cygwin_ansvr\usr\share\astrometry\data)

Once the plate solver has been installed, along with the index files, launch Tycho and choose "*Action->Plate Solve Images*". Provide the path to the `cygwin_ansvr` directory, this is usually going to reside in your local `AppData` directory.

If you know the plate scale of your imaging chip, you can provide a lower- and upper-bound on the pixel 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.

"Downsample=2" will usually produce a good result. However, if you have a wide field or a large image then "Downsample=4" is probably better, and a lot faster. If you are experiencing plate solve failures, you might experiment with different values to see what works best for your images.

The option "Automatically perform second attempt with different source image" allows the plate solver to automatically perform a second attempt if the first attempt fails. It will also select the middle image in the sequence as the source image and, upon successful solve, copy its WCS information over to the first image in the sequence. This can be a useful fallback if the first image fails to solve.

The option "Narrow search to RA/Dec in FITS header" is useful to prevent false matches from occurring. It is very rare for the plate solver to produce a false match, but it has happened on at least one occasion. Therefore, this option can be enabled to instruct the plate solver to search within a given radius of the

RA/Dec specified within the FITS header of the first image. If the RA/Dec is not found in the FITS header, the plate solver will revert to conducting a "blind search", which is the same behavior as if the option were disabled. The default search radius is 15.00 degrees, meaning that the RA/Dec in the FITS header can be off by that amount from the actual RA/Dec. Evaluation of numerous datasets has shown that while most FITS headers will have the RA/Dec correct to within 5 arcmin, some can be off by several degrees. Thus, to ensure that the plate solver is not overly constrained, it is advisable to have a sufficient radius, which will still reduce the probability of an incorrect solution being returned.

Plate Solve Option 3: Using the Astrometry.net Service

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.

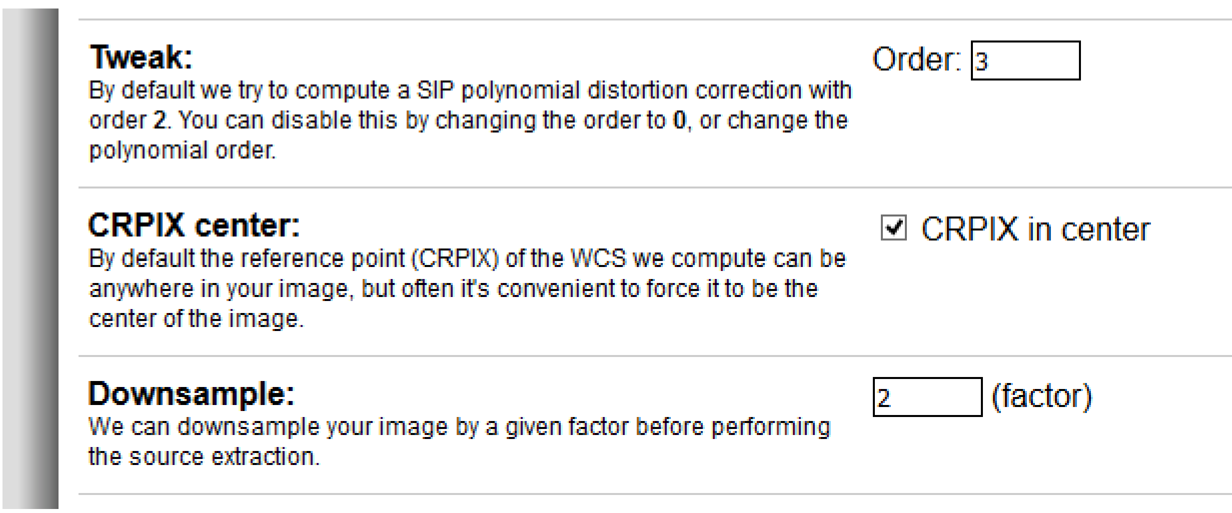


Figure 14 - Advanced Settings for Online Plate Solver

"Tweak" should be set to 3 in order to satisfy the 3^{rd}-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 15.

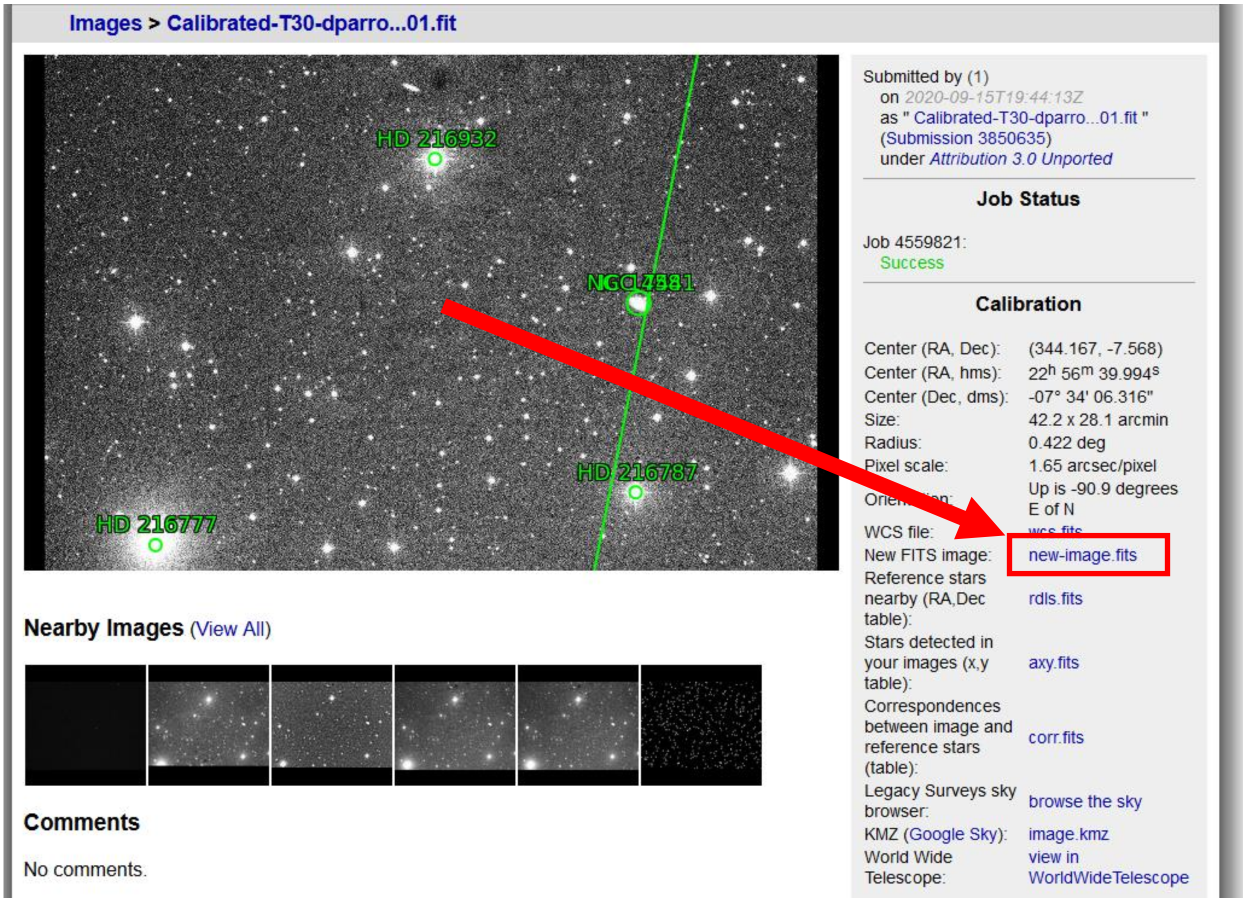


Figure 15 - Retrieving the Plate Solved Image

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, the astrometry.net service is a bit more tedious than using the integrated solvers, but is 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. For the purpose of providing a tutorial, some steps of the process are outlined.

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

From section 16, "How do I begin?":

> If your site does not have an observatory code, one will be assigned upon acceptance of your initial submission. Your initial submission should contain at least six numbered minor planets each on pairs of nearby nights as well as one numbered Near-Earth object observed on two distinct nights. If weather interferes, the two nights can be some weeks apart. Report at least three observations of each object from each night: do not report single positions per night. Batches that contain single positions will be returned in their entirety to the submitter. We will check these positions and advise you on their quality. As a general rule we advise you NOT to observe very low-numbered objects--e.g., (1), (2), (51) and very bright objects. In your initial batch please submit astrometry of objects fainter than 14th magnitude. In addition, you should try and observe objects of various brightness.

From someone who recently acquired an MPC code:

> Your first submission of measurements should contain XXX for observatory code for the MPC to recognize it as a code request/application. Immediately after you make your first XXX submission, you will get an email directing you to enter some information for your code application. That submission and all other submissions you make until you get a code should only contain two-night measurements on asteroids in the range identifier of 800-40,000.

From another user who also recently acquired an MPC code:

> After my initial submission was accepted (with observations of two different asteroids) I was asked to submit more observations of other asteroids. The email also had a link to fill in coordinates of the observatory and some other details. For the second set of observations, I used code XXX and sent observations of 4 asteroids. Within a few days, I sent observations of one more asteroid with code XXX. I waited for two weeks from the time I submitted my second set to get my observatory code.

The Tycho software makes it easy to generate a report in the format required by the MPC. This is explained later as you work through the example datasets. It is also recommended to use the MPC1992 format for the initial submissions, and later the ADES format if desired. This is because ADES does not yet support the "XXX" (new observatory) code.

Until you have a code, you 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.

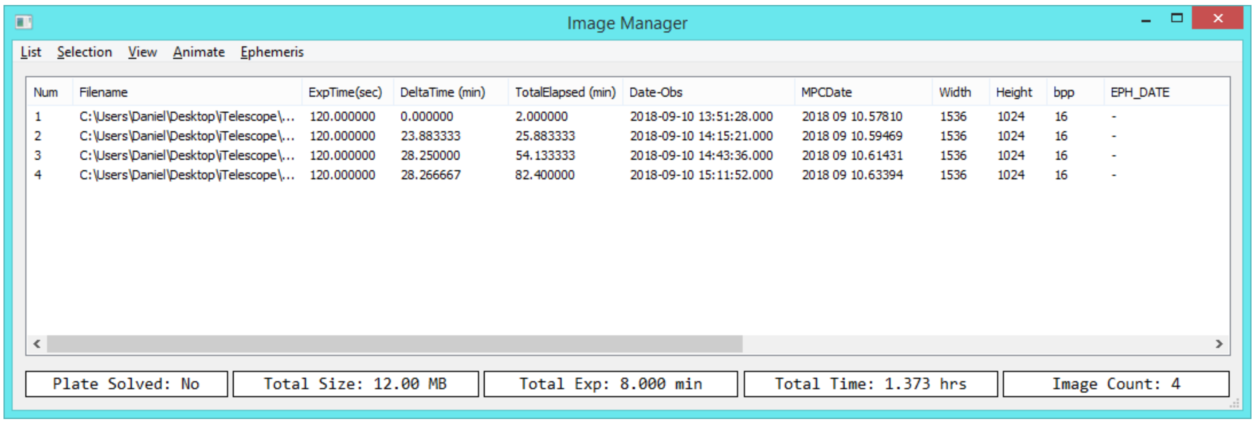


Figure 16 - Image Manager with Four Images Loaded

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. For flat frame calibration, “pseudo flat” works quite well, especially 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.

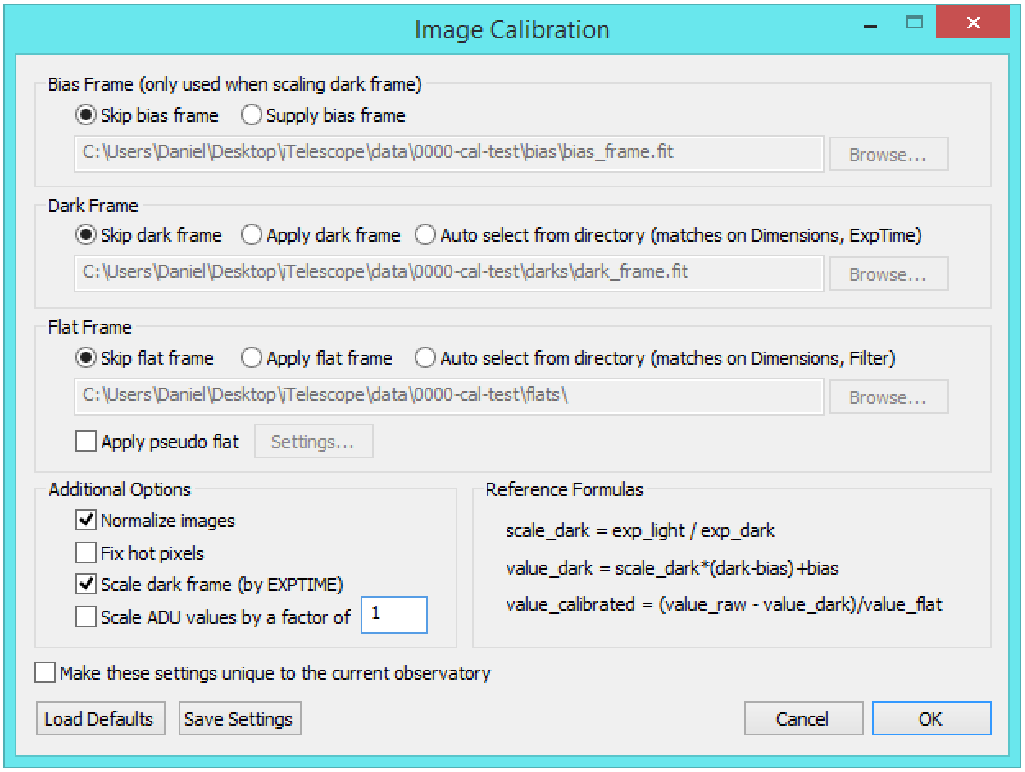


Figure 17 - Calibration Settings

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.

Step 4: Align the Images

The next step is to align the images. Proceed to *Action->Align Images* and choose the desired alignment option. If you have not already done so, navigate to the relevant tab and configure the settings. Click “OK” to proceed. As before, the images will be saved to a new directory, this time labeled “ds1_c_a”.

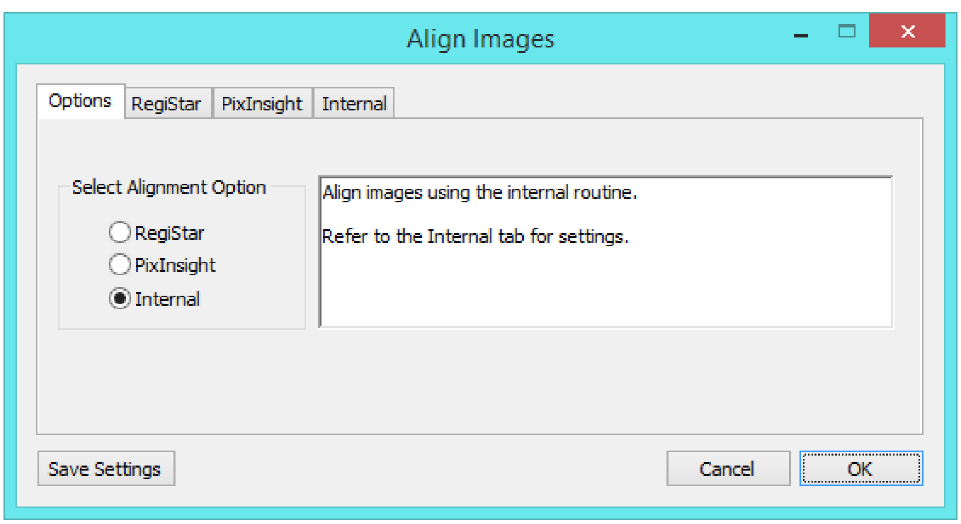


Figure 18 - Align Images

Step 5: Plate Solve the Images

Navigate to the newly-aligned images, contained in directory "ds1_c_a". Load them as before. Now go to *Action->Plate Solve Images*.

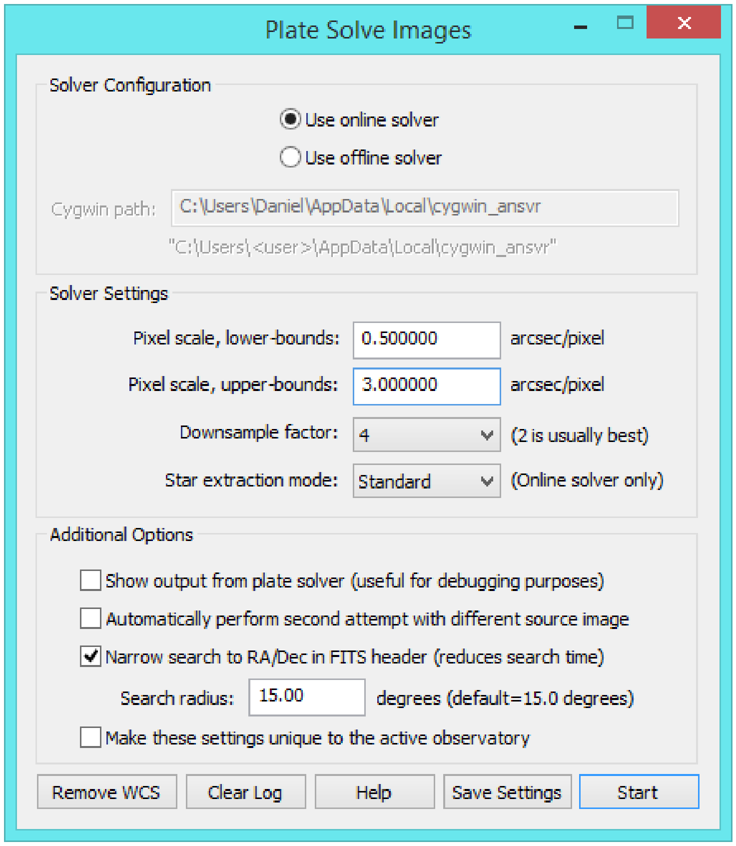


Figure 19 - Plate Solve Images

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 19. For more details, refer to the section labeled "Configure the Plate Solver". Proceed to plate solve the images by clicking the "Start" button.

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 "Image Viewer".

If you do not see any stars, make sure *Display->Catalog Stars* is checked, 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.

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.

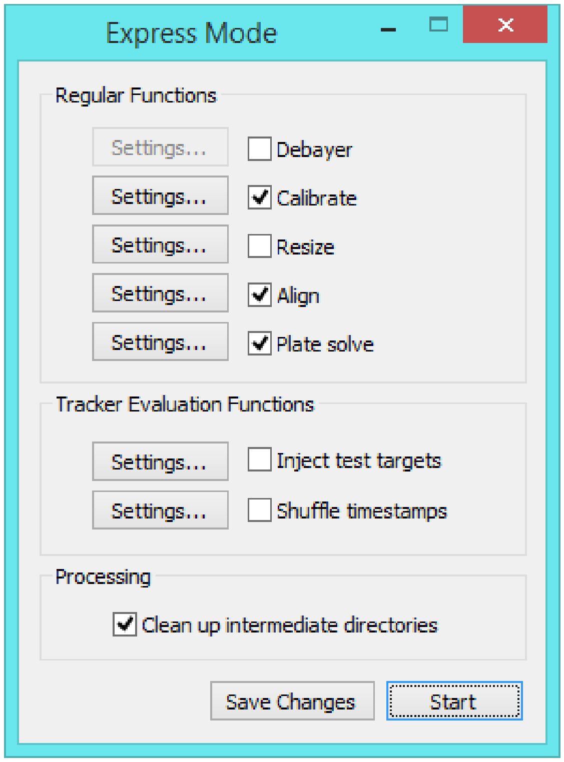


Figure 20 - Express Mode

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, aligned, and plate solved, it is time to start generating some measurements (also known as “observations”). Load up 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 21. Then click the “Next” button and make sure to use the same settings as shown in Figure 22. Then, click “Finished”.

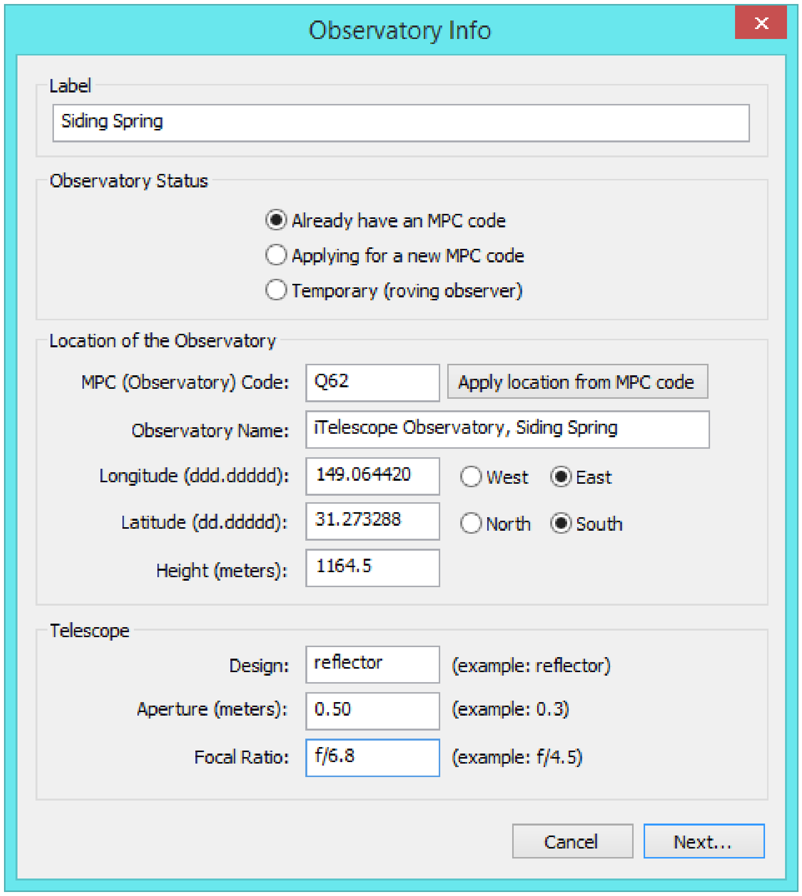


Figure 21 - Adding Observatory, Page 1/2

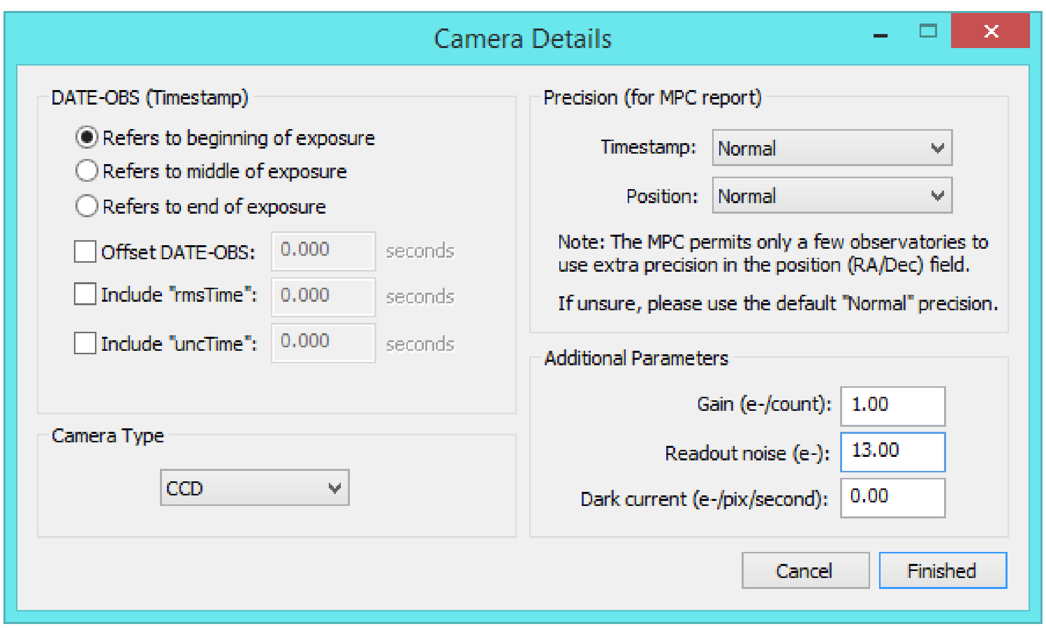
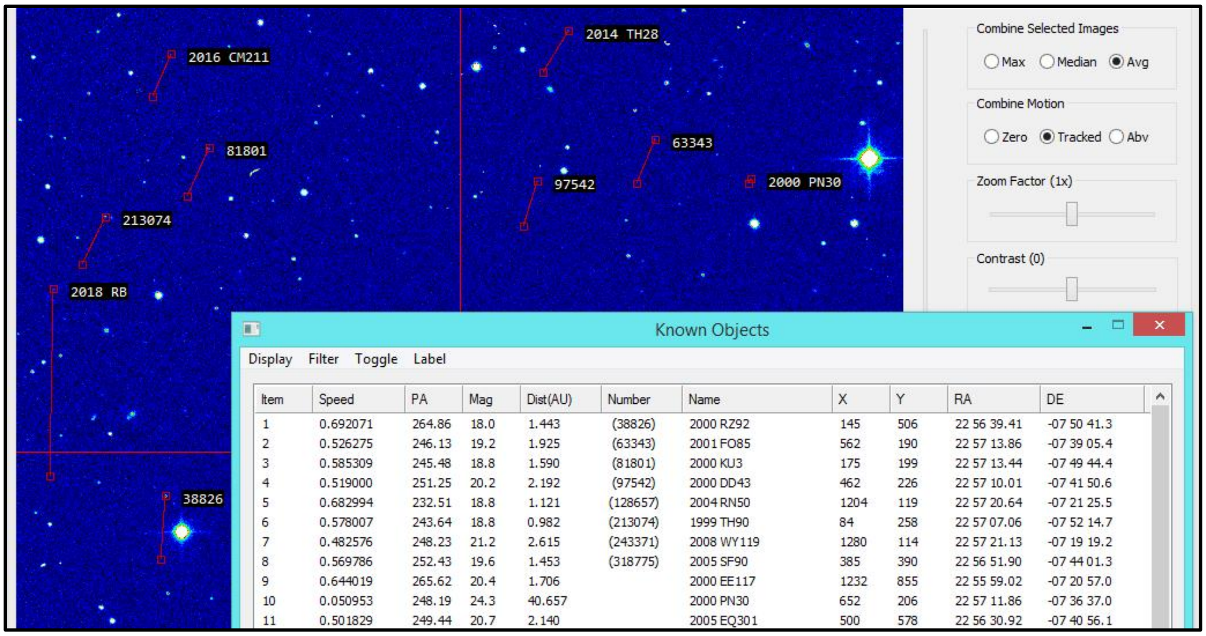


Figure 22 - Adding Observatory, Page 2/2

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 the *Find_Orb* software installed and configured in order to precisely compute the positions of asteroids, so if you have not yet configured the settings for *Find_Orb*, it is recommended to do so now. Refer to the section labeled “Configure the Find_Orb Software” for more details.



Item	Speed	PA	Mag	Dist(AU)	Number	Name	X	Y	RA	DE
1	0.692071	264.86	18.0	1.443	(38826)	2000 RZ92	145	506	22 56 39.41	-07 50 41.3
2	0.526275	246.13	19.2	1.925	(63343)	2001 FO85	562	190	22 57 13.86	-07 39 05.4
3	0.585309	245.48	18.8	1.590	(81801)	2000 KU3	175	199	22 57 13.44	-07 49 44.4
4	0.519000	251.25	20.2	2.192	(97542)	2000 DD43	462	226	22 57 10.01	-07 41 50.6
5	0.682994	232.51	18.8	1.121	(128657)	2004 RN50	1204	119	22 57 20.64	-07 21 25.5
6	0.578007	243.64	18.8	0.982	(213074)	1999 TH90	84	258	22 57 07.06	-07 52 14.7
7	0.482576	248.23	21.2	2.615	(243371)	2008 WY119	1280	114	22 57 21.13	-07 19 19.2
8	0.569786	252.43	19.6	1.453	(318775)	2005 SF90	385	390	22 56 51.90	-07 44 01.3
9	0.644019	265.62	20.4	1.706		2000 EE117	1232	855	22 55 59.02	-07 20 57.0
10	0.050953	248.19	24.3	40.657		2000 PN30	652	206	22 57 11.86	-07 36 37.0
11	0.501829	249.44	20.7	2.140		2005 EQ301	500	578	22 56 30.92	-07 40 56.1

Figure 23 - 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 24.

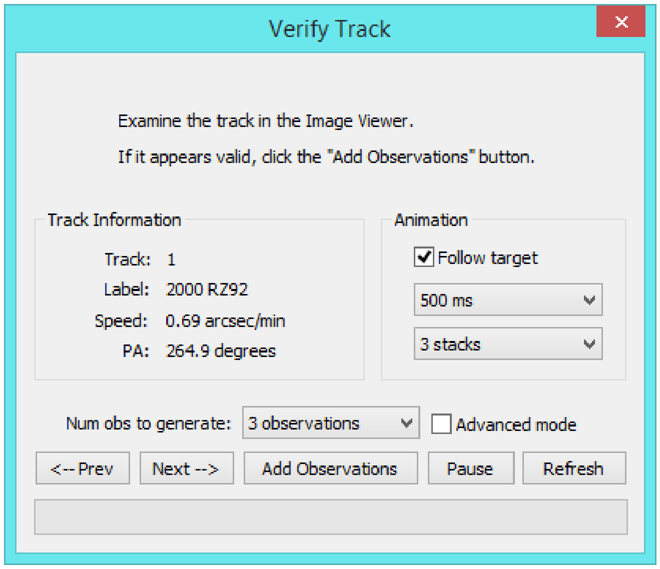


Figure 24 - Verify Track

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.

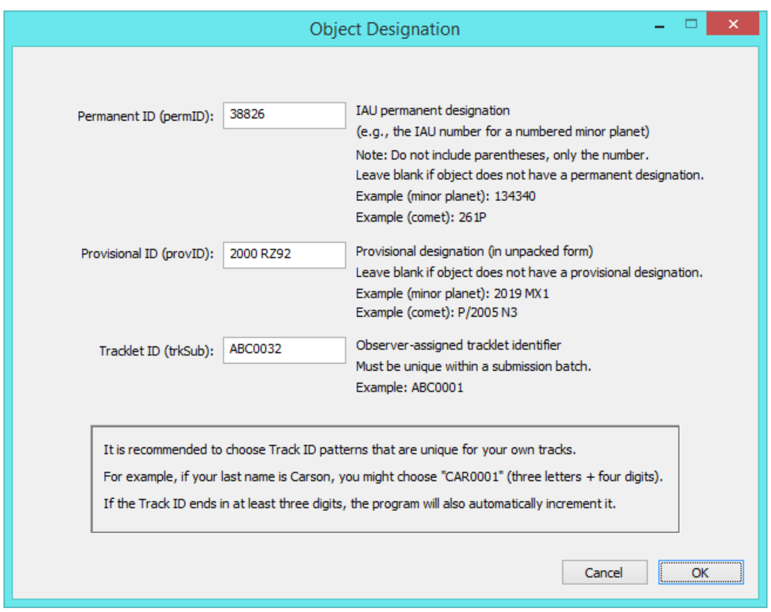


Figure 25 - Object Designation

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.

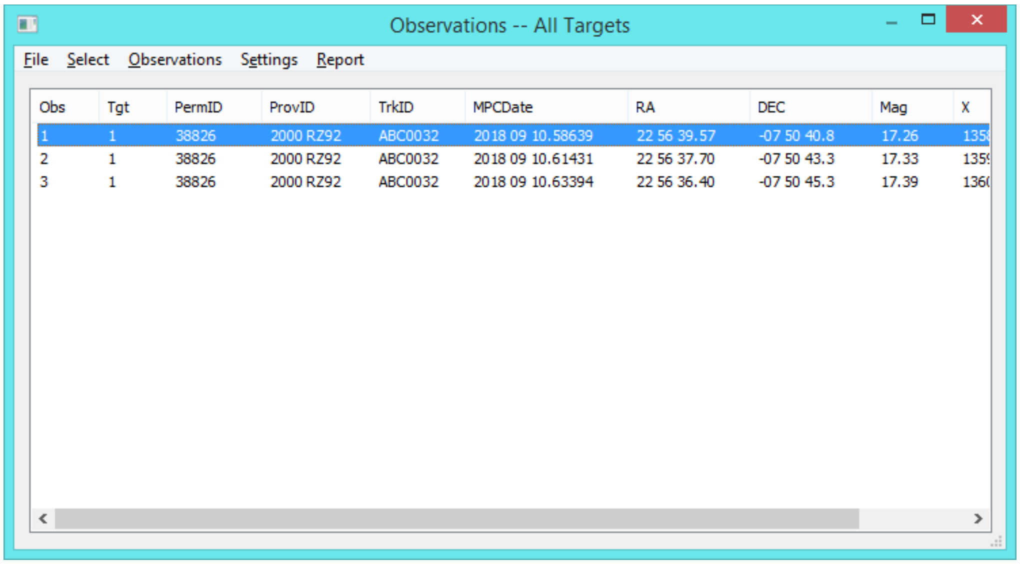


Figure 26 - Observations -- All Targets

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 last two images.

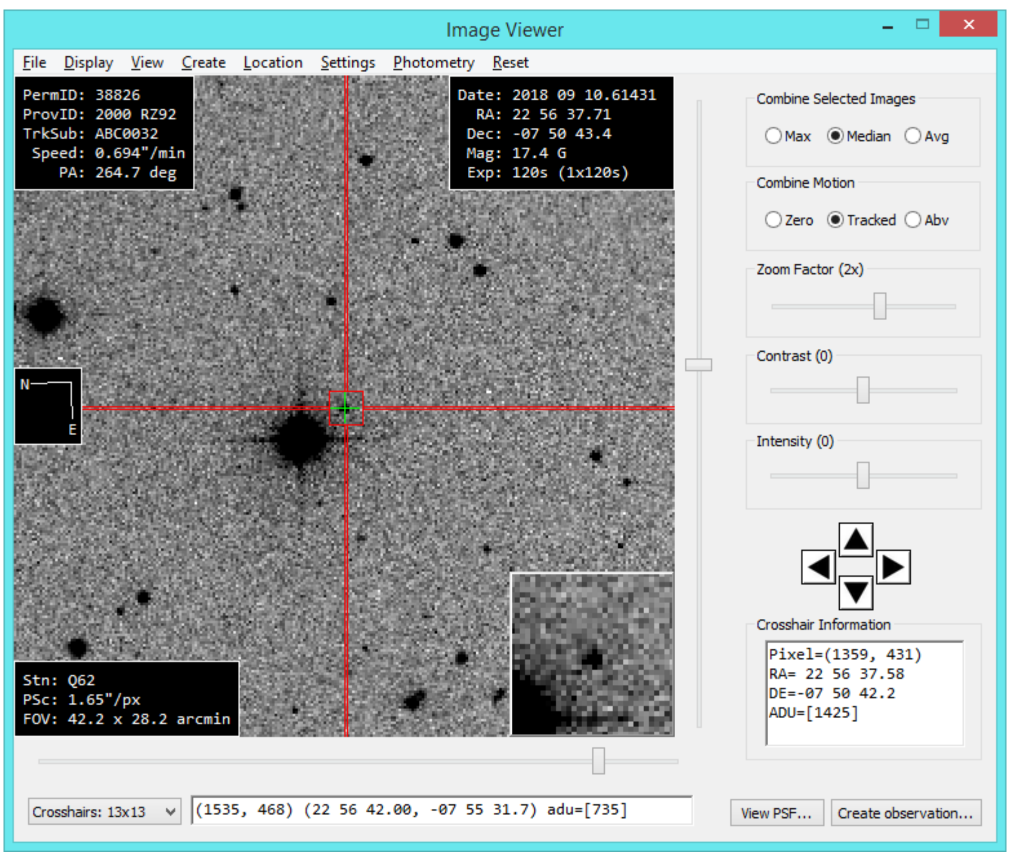


Figure 27 - Visual Inspection of Observations

Step 10: Validating the Measurements

You can validate the measurements by selecting them, then right-click and choose "View with Existing 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 "View in Find_Orb" (the button in the lower-right corner of the window), and the *Find_Orb* software will show the residuals for these three measurements that you just generated.

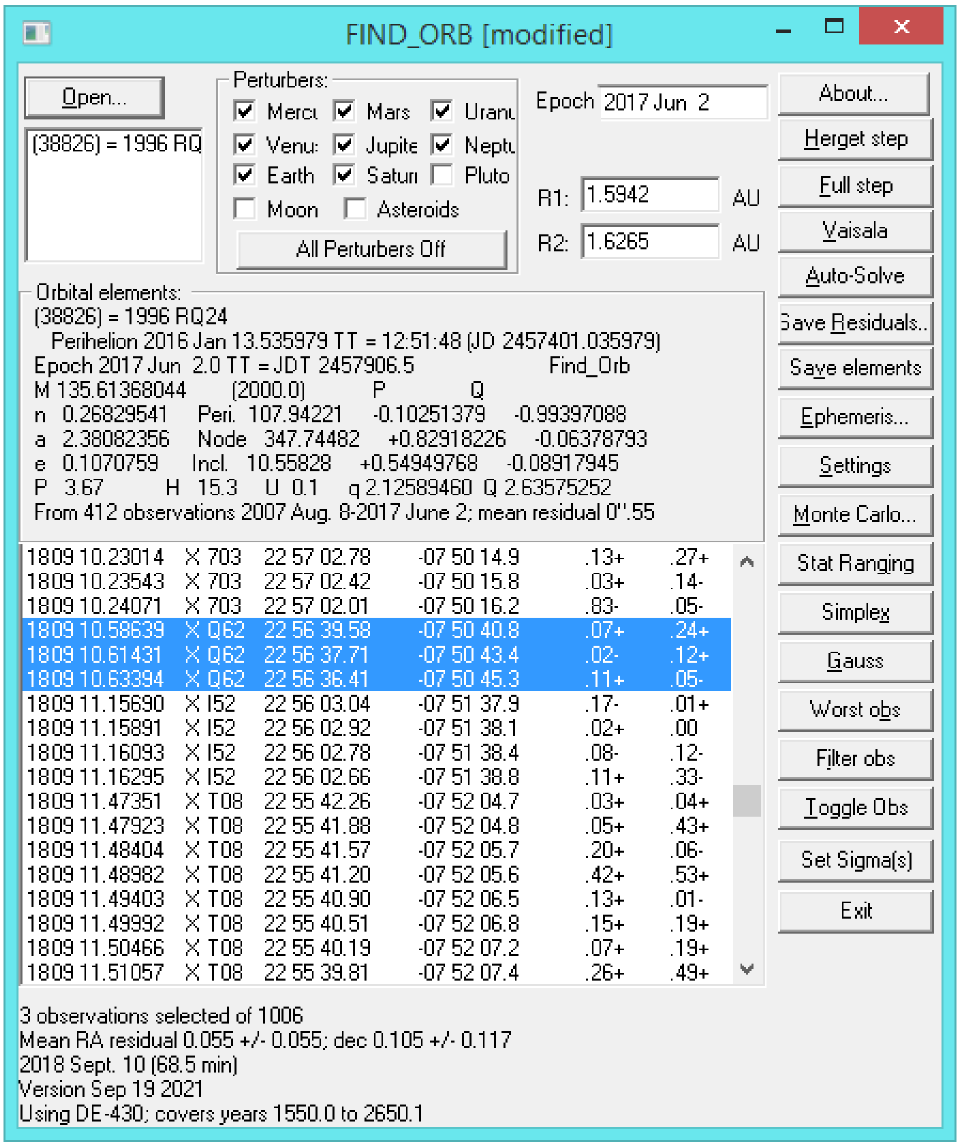


Figure 28 - FindOrb Results

As you can see, the measurements are quite good, with “0.31” 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*.

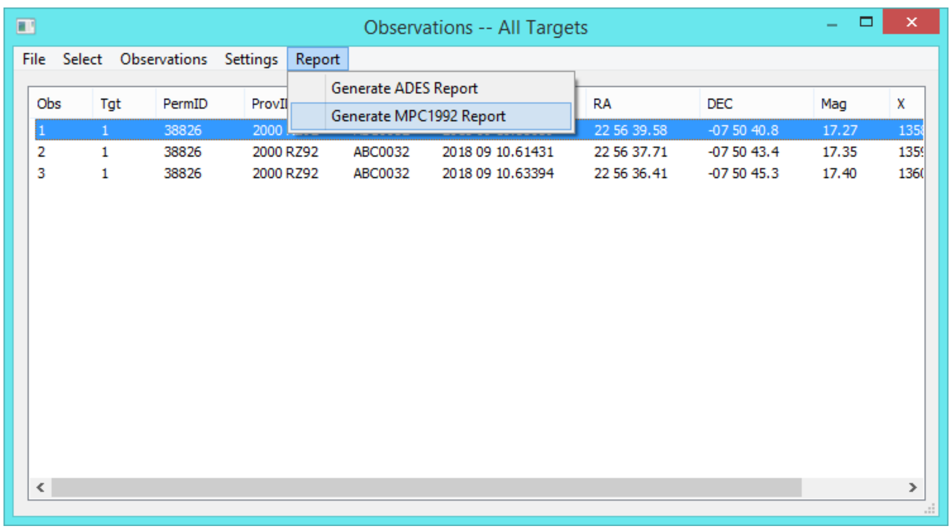


Figure 30 - Generating the Measurement Report

```
COD Q62
CON D. Parrott
OBS D. Parrott
MEA D. Parrott
TEL 500mm Reflector + CCD
NUM 3
ACK MPCReport file updated 2021.01.03 18:26:22
NET Gaia DR2
38826         KC2018 09 10.58639 22 56 39.58 -07 50 40.8          17.3 G      Q62
38826          C2018 09 10.61431 22 56 37.71 -07 50 43.4          17.4 G      Q62
38826          C2018 09 10.63394 22 56 36.41 -07 50 45.3          17.4 G      Q62
----- end -----
```

Figure 29 - 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, align, and plate solve 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 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, align, and plate solve 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

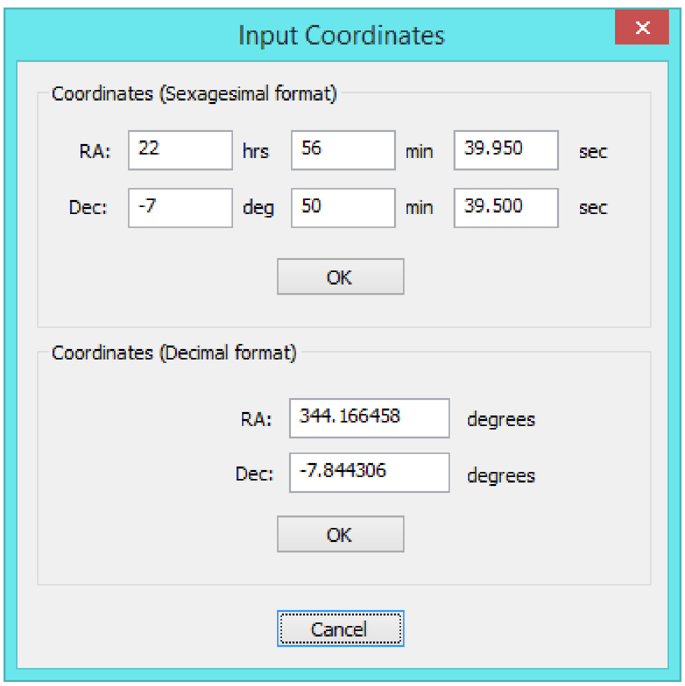


Figure 31 - Input RA/Dec Coordinates

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 39.95
DE = -07 50 39.50 (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.

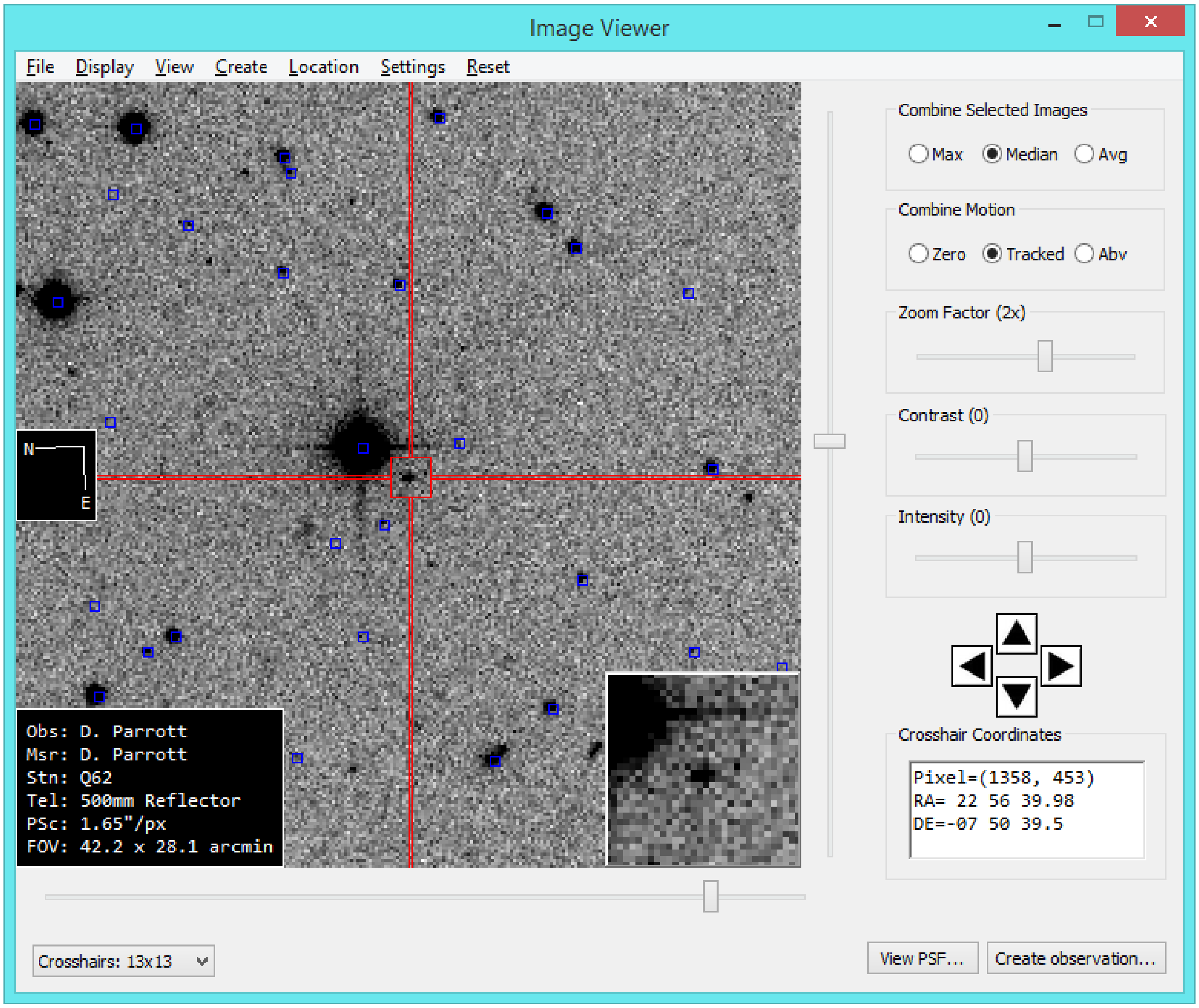


Figure 32 - Asteroid (38826)

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. 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”.

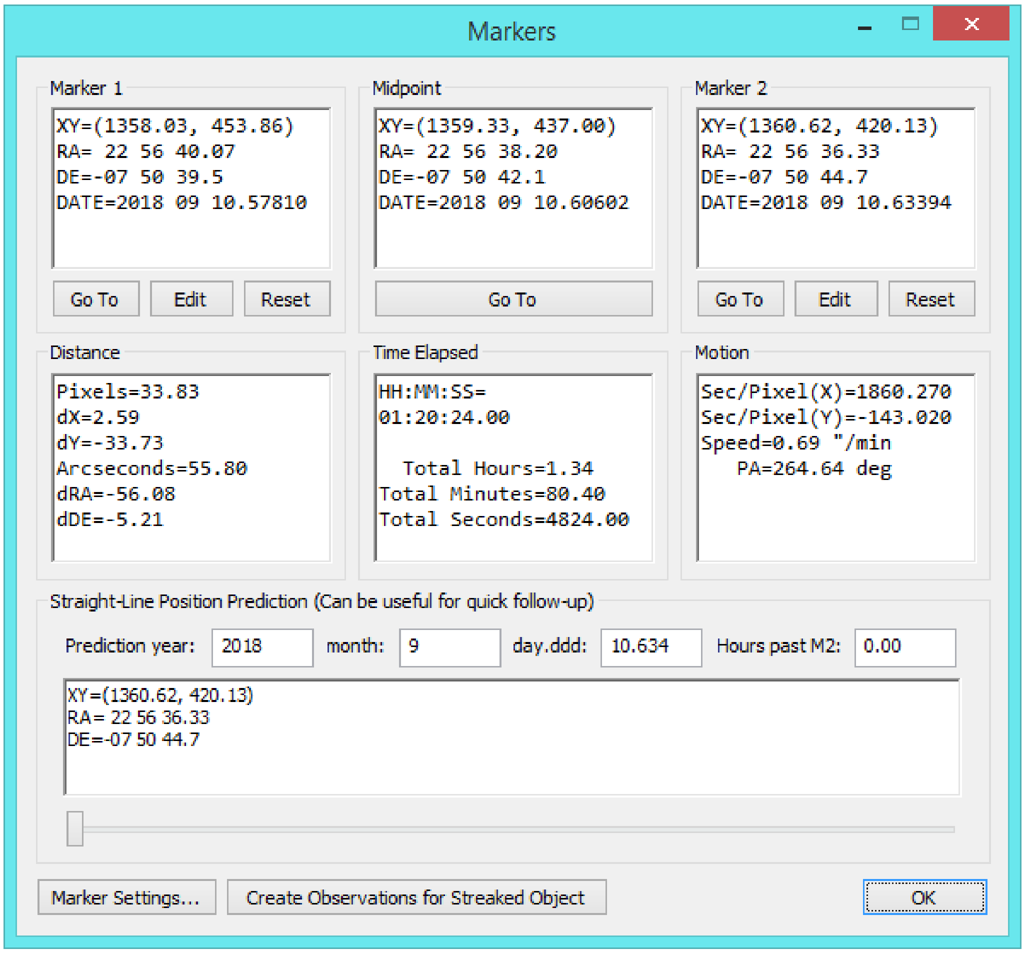


Figure 33 - Markers Window

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.

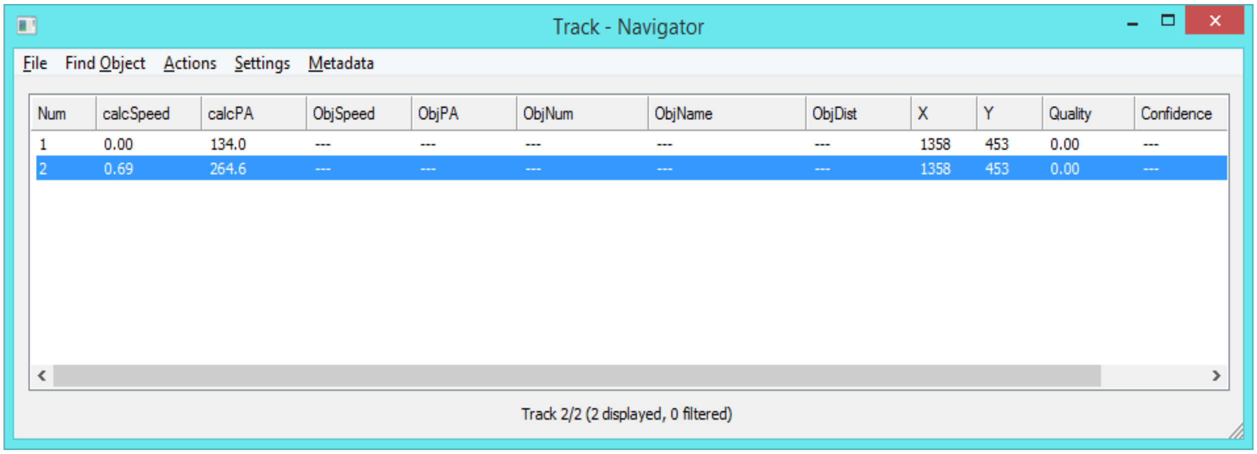


Figure 34 - Manually Created Tracks

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.

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", "avg" (mean), 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 "Avg" 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 "Avg" 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.

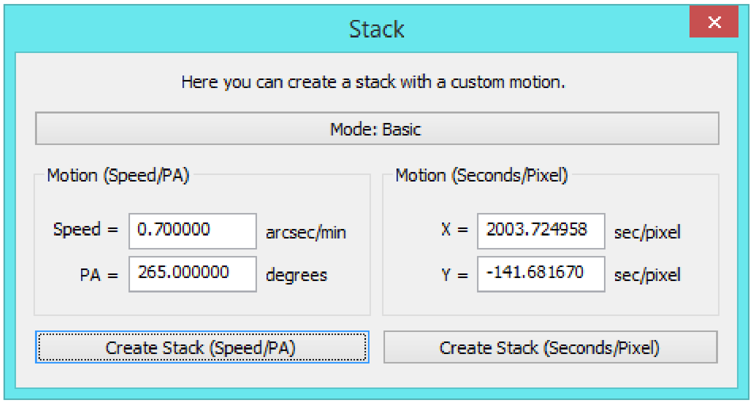


Figure 35 - Create Stack (Custom)

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 36.

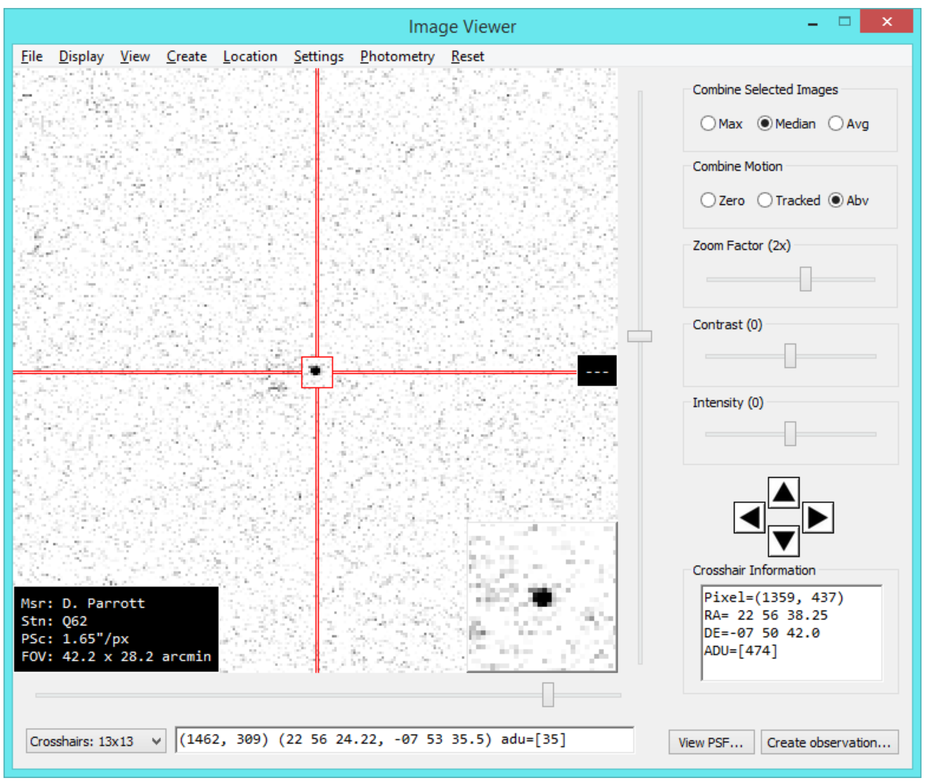


Figure 36 - "Abv" Stack

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).

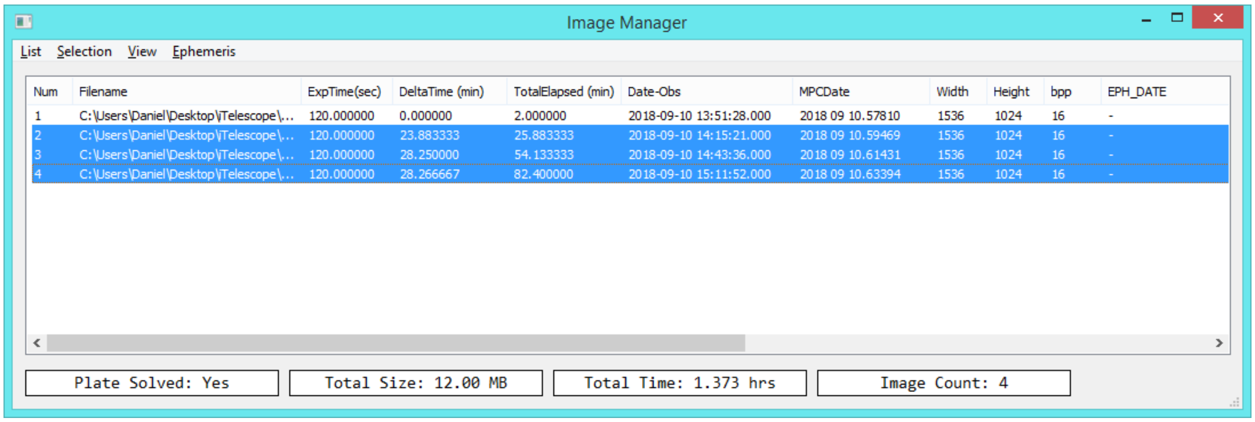


Figure 37 - Selecting the Last 3 Images

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 38, when the stack is created with "Avg" layer (not Abv), it is evident that three images were stacked because each star leaves behind three "footprints" in the image.

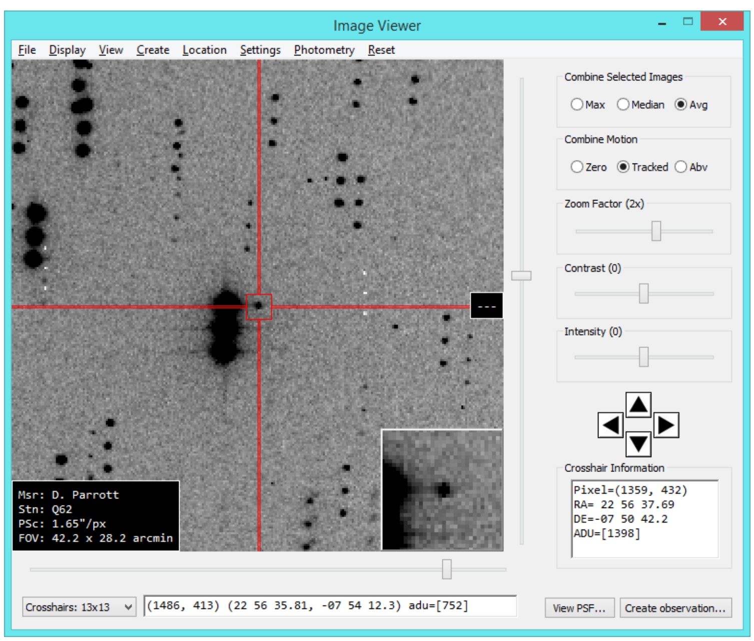


Figure 38 - Stack with 3 Images

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.

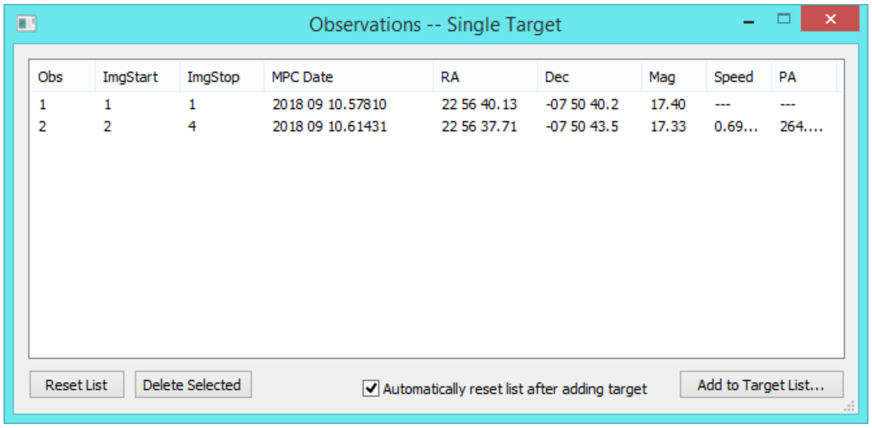


Figure 39 - Observations -- Single Target

Tip: In a later example, I will also show how to create these “sub-stacks” using the “Track – Positions” window, which is far 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 41.

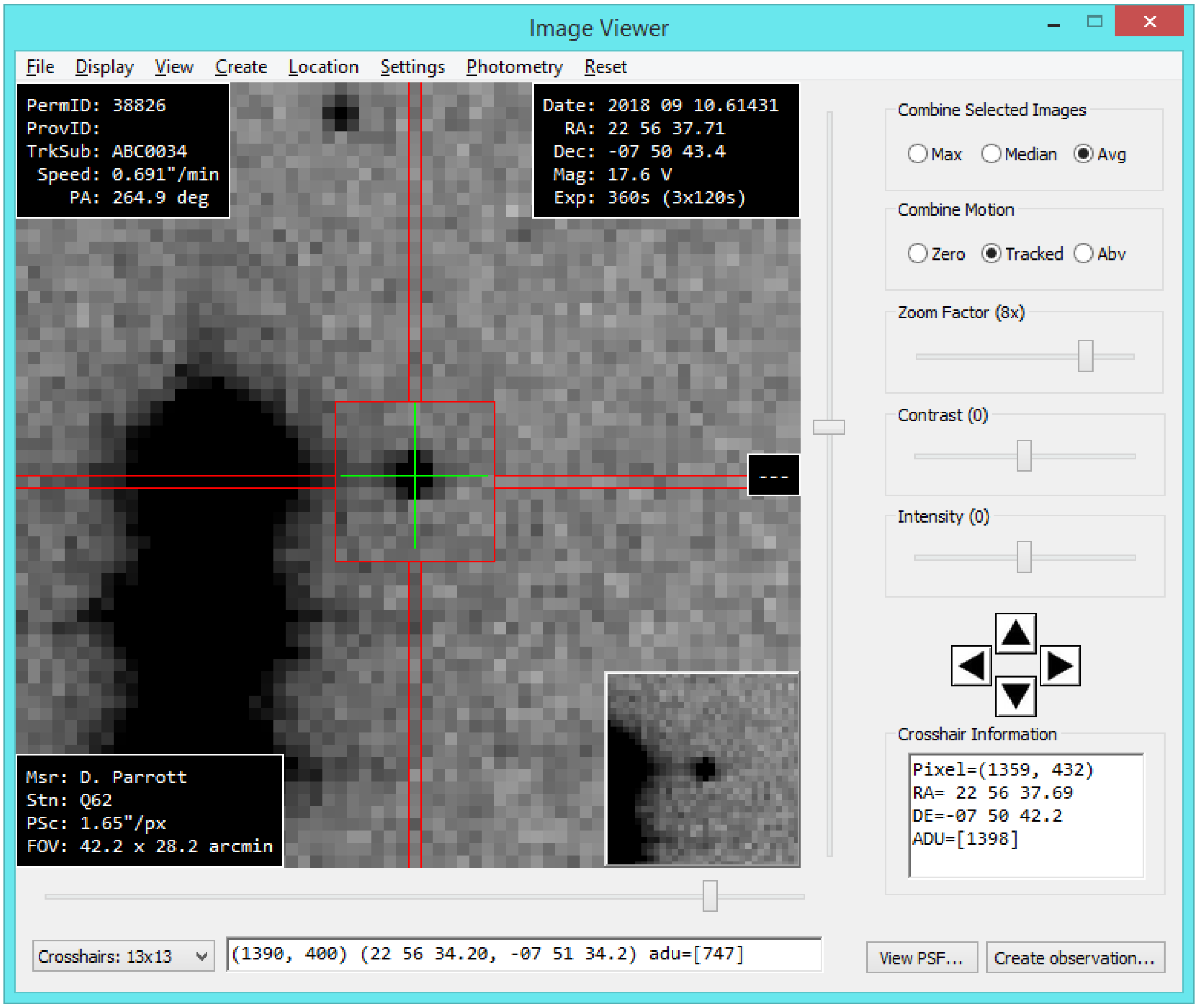


Figure 40 - Green Centroid Indicator

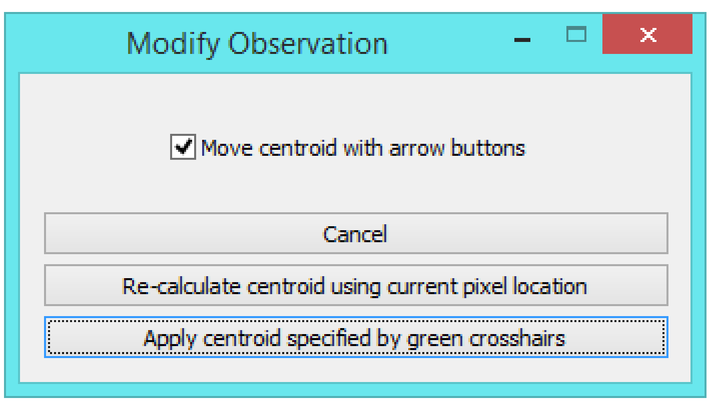


Figure 41 - Modify Observation

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 “Avg” 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, align, and plate solve 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, alignment, and plate solving. 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: View the Observations in Find_Orb

Click the button in the lower-right corner, labeled “View in FindOrb”. The *Find_Orb* program will launch and compute the orbit for this object using the observations from the text form. Continue to leave *Find_Orb* running in the background for the next step.

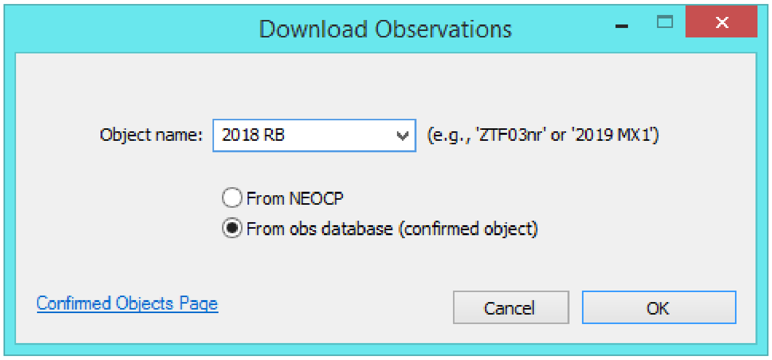


Figure 42 - Download Observations

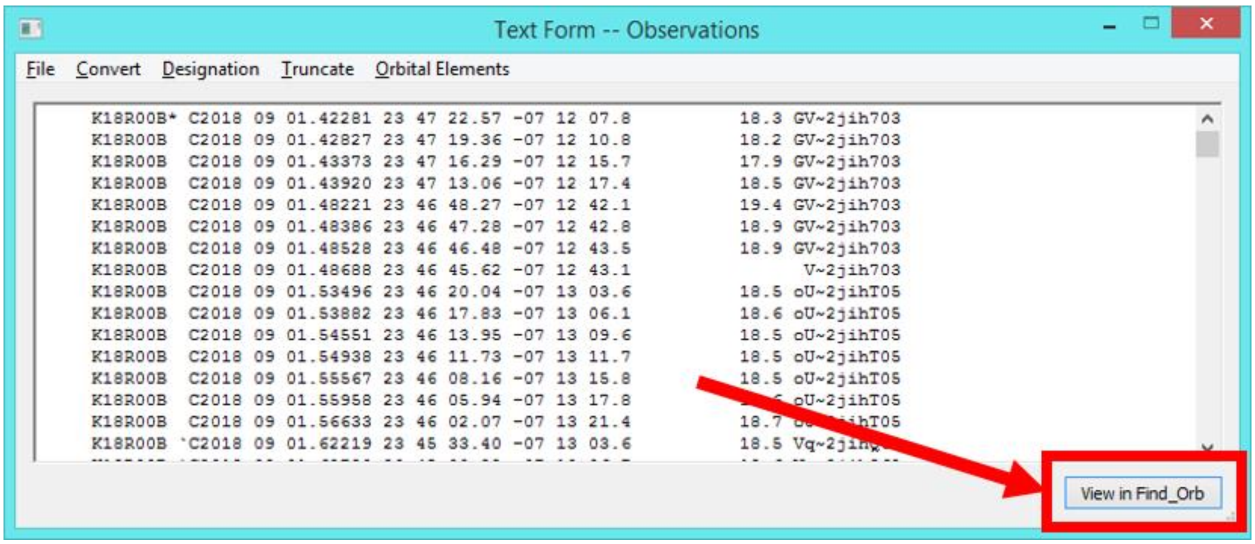


Figure 43 - Text Form -- Observations

Step 4: Attach Ephemeris to the Dataset

With Find_Orb still in the background, navigate back to the "Image Manager" and choose *Ephemeris->Attach to Dataset* from the "Image Manager" menu. From the window that appears, double-click on the appropriate instance of *Find_Orb* (if you have multiple instances running, choose the one with "2018 RB"). 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 a display of a stacked image according to the computed motion of the object.

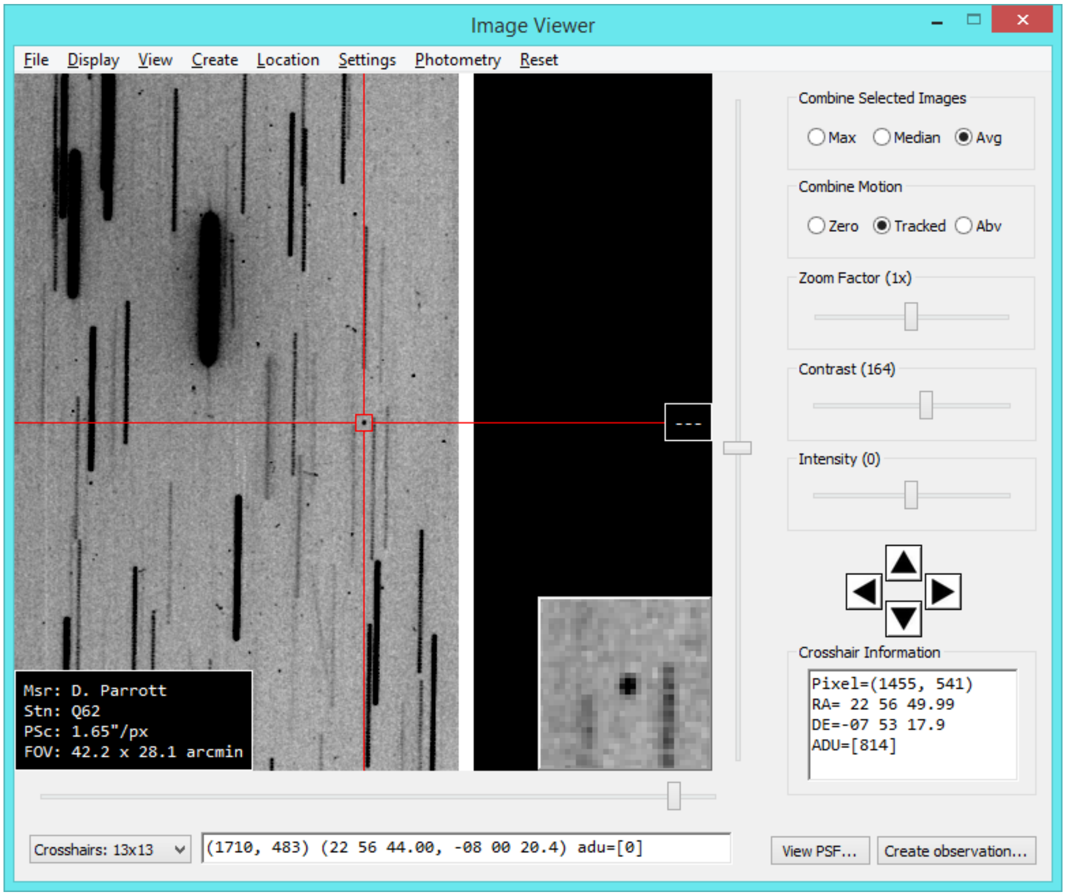


Figure 44 - Result of "Create Stack - Ephemeris"

In Figure 44, 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 “Avg”, 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 (164) for the contrast slider.

Step 6: Create a Track

Double-click the object to ensure that it is centered in the crosshairs. 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 Existing 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 “View in Find_Orb” and you should see the residuals (errors) for the observations you just created. From this instance of *Find_Orb*, it appears that the highest residual is “0.21”, 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.

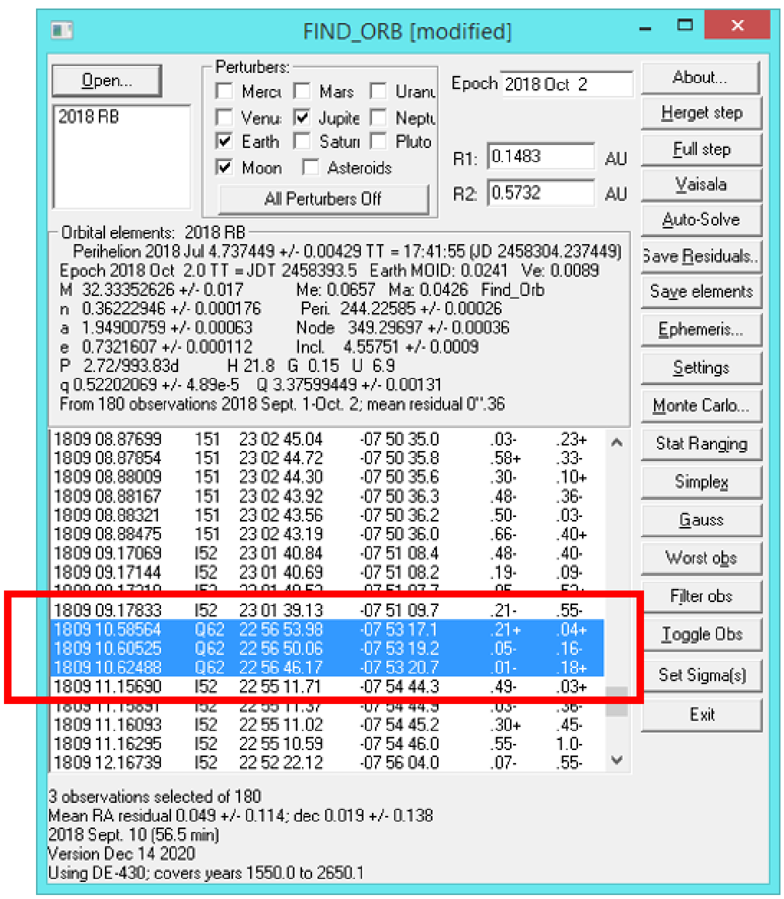


Figure 45 - Validating the Measurements

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. If you want to bring it to the front, you can go to *Window – Blink Stacks* from the main menu.

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, align, and plate solve). 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 sensitivity.

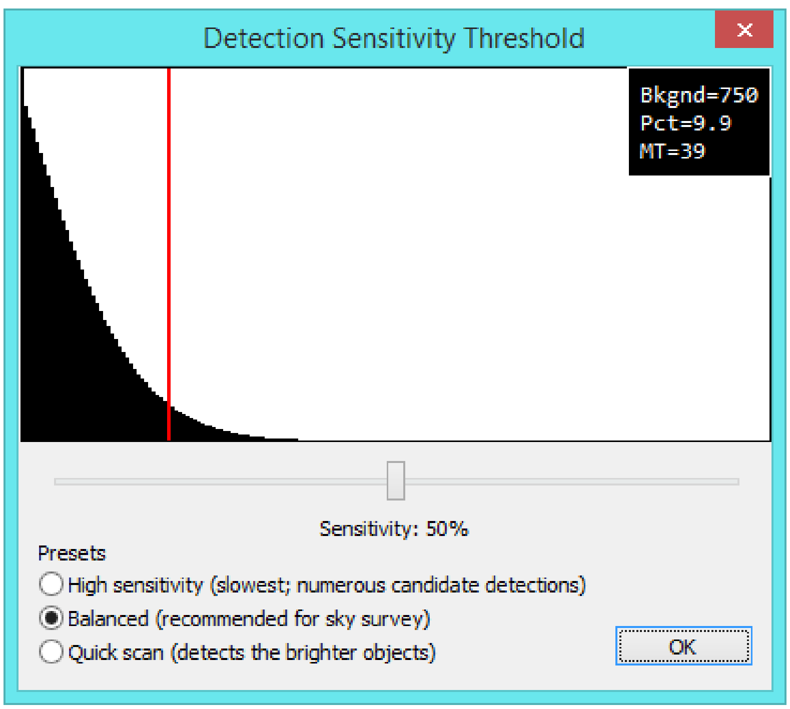


Figure 46 - Sensitivity Threshold

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 47.

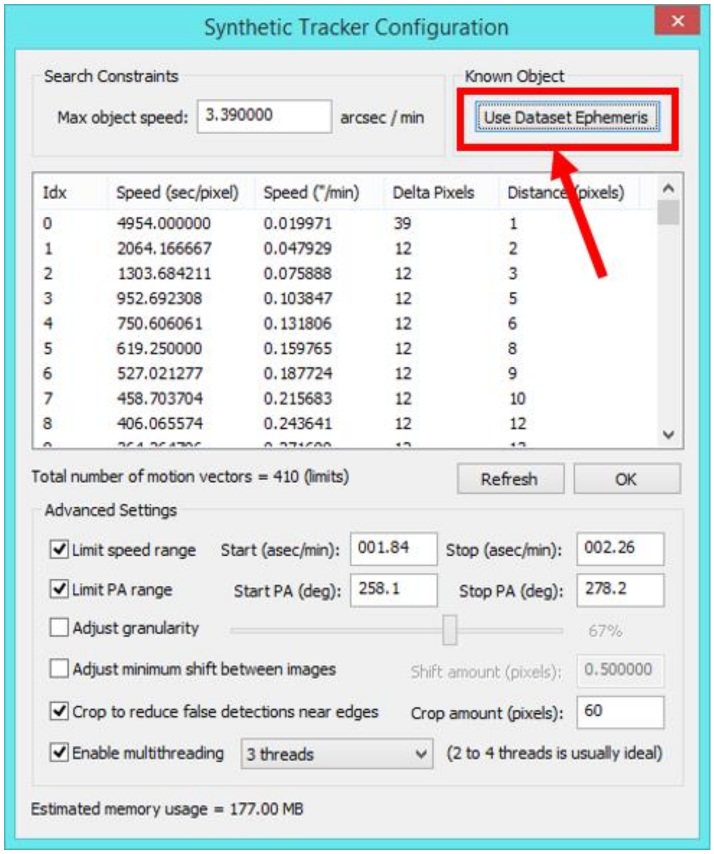


Figure 47 - Synthetic Tracker Configuration

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 MPCORB.CUSTOM* from the "Text Form – Observations" window. This invokes the *Find_Orb* program 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 will 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. 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->Modify MPCORB.CUSTOM*. Then click on the item and press the "Delete" key, or right-click and choose "Delete" from the popup menu. Then save the changes by going to *File->Save* from the menu of the "Orbital Elements [MPCORB.CUSTOM" window.

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 system with an NVIDIA RTX 2080, it took 19 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, alignment, and plate solving. As usual, if you need a refresher on how to perform these steps, consult 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.

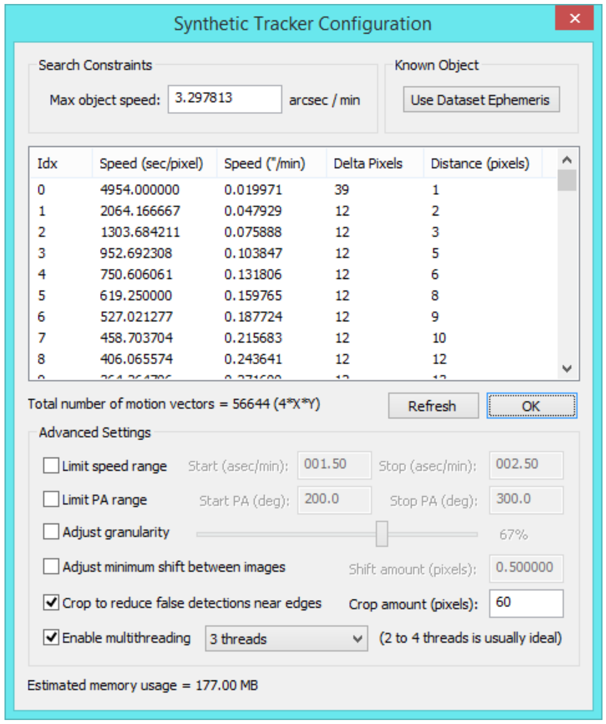


Figure 48 - Synthetic Tracker Configuration

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.

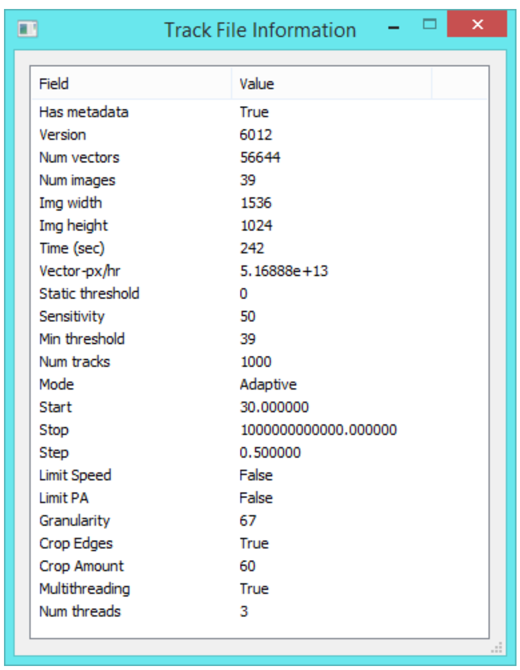
Finally, you may want to enable multithreading to speed up the processing. In CPU mode, this setting overrides the setting in the Device Selector, and the threading works by dividing up each image for processing. In GPU mode, this setting works by having each thread operate on its own motion vector, 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 48. 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 system using an NVIDIA RTX 2080 graphics card, the total time to process was 242 seconds.

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.


Track File Information

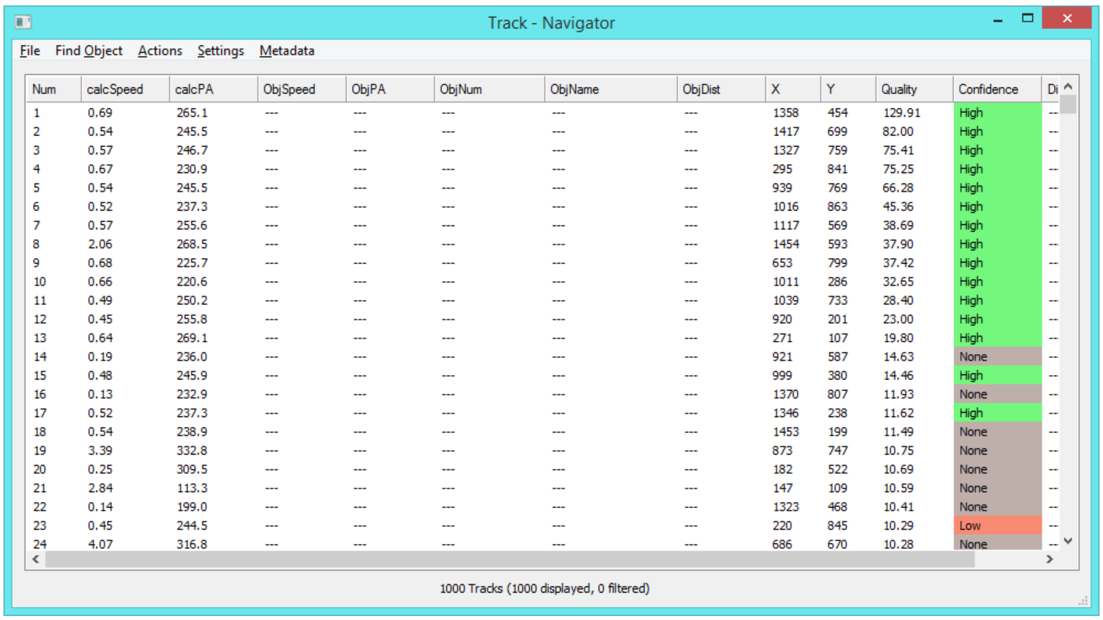
Field	Value
Has metadata	True
Version	6012
Num vectors	56644
Num images	39
Img width	1536
Img height	1024
Time (sec)	242
Vector-px/hr	5.16888e+13
Static threshold	0
Sensitivity	50
Min threshold	39
Num tracks	1000
Mode	Adaptive
Start	30.000000
Stop	1000000000000.000000
Step	0.500000
Limit Speed	False
Limit PA	False
Granularity	67
Crop Edges	True
Crop Amount	60
Multithreading	True
Num threads	3

Figure 49 - 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	ObjNum	ObjName	ObjDist	X	Y	Quality	Confidence	Di
1	0.69	265.1	---	---	---	---	---	1358	454	129.91	High	--
2	0.54	245.5	---	---	---	---	---	1417	699	82.00	High	--
3	0.57	246.7	---	---	---	---	---	1327	759	75.41	High	--
4	0.67	230.9	---	---	---	---	---	295	841	75.25	High	--
5	0.54	245.5	---	---	---	---	---	939	769	66.28	High	--
6	0.52	237.3	---	---	---	---	---	1016	863	45.36	High	--
7	0.57	255.6	---	---	---	---	---	1117	569	38.69	High	--
8	2.06	268.5	---	---	---	---	---	1454	593	37.90	High	--
9	0.68	225.7	---	---	---	---	---	653	799	37.42	High	--
10	0.66	220.6	---	---	---	---	---	1011	286	32.65	High	--
11	0.49	250.2	---	---	---	---	---	1039	733	28.40	High	--
12	0.45	255.8	---	---	---	---	---	920	201	23.00	High	--
13	0.64	269.1	---	---	---	---	---	271	107	19.80	High	--
14	0.19	236.0	---	---	---	---	---	921	587	14.63	None	--
15	0.48	245.9	---	---	---	---	---	999	380	14.46	High	--
16	0.13	232.9	---	---	---	---	---	1370	807	11.93	None	--
17	0.52	237.3	---	---	---	---	---	1346	238	11.62	High	--
18	0.54	238.9	---	---	---	---	---	1453	199	11.49	None	--
19	3.39	332.8	---	---	---	---	---	873	747	10.75	None	--
20	0.25	309.5	---	---	---	---	---	182	522	10.69	None	--
21	2.84	113.3	---	---	---	---	---	147	109	10.59	None	--
22	0.14	199.0	---	---	---	---	---	1323	468	10.41	None	--
23	0.45	244.5	---	---	---	---	---	220	845	10.29	Low	--
24	4.07	316.8	---	---	---	---	---	686	670	10.28	None	--

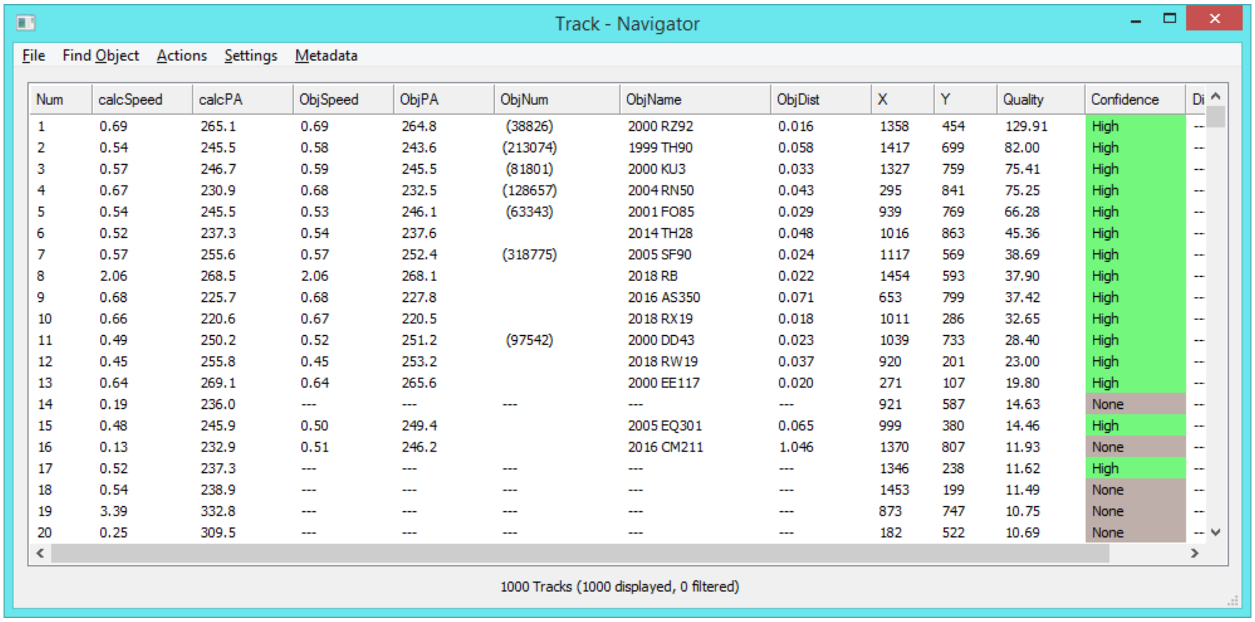
Figure 50 - Track Results

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 16 -- is an erroneous match, having an "ObjDist" value exceeding 1.0 arcminute, while the other matches have an "ObjDist" value under 0.3 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 #17 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	ObjNum	ObjName	ObjDist	X	Y	Quality	Confidence	Di
1	0.69	265.1	0.69	264.8	(38826)	2000 RZ92	0.016	1358	454	129.91	High	--
2	0.54	245.5	0.58	243.6	(213074)	1999 TH90	0.058	1417	699	82.00	High	--
3	0.57	246.7	0.59	245.5	(81801)	2000 KU3	0.033	1327	759	75.41	High	--
4	0.67	230.9	0.68	232.5	(128657)	2004 RN50	0.043	295	841	75.25	High	--
5	0.54	245.5	0.53	246.1	(63343)	2001 FO85	0.029	939	769	66.28	High	--
6	0.52	237.3	0.54	237.6		2014 TH28	0.048	1016	863	45.36	High	--
7	0.57	255.6	0.57	252.4	(318775)	2005 SF90	0.024	1117	569	38.69	High	--
8	2.06	268.5	2.06	268.1		2018 RB	0.022	1454	593	37.90	High	--
9	0.68	225.7	0.68	227.8		2016 AS350	0.071	653	799	37.42	High	--
10	0.66	220.6	0.67	220.5		2018 RX19	0.018	1011	286	32.65	High	--
11	0.49	250.2	0.52	251.2	(97542)	2000 DD43	0.023	1039	733	28.40	High	--
12	0.45	255.8	0.45	253.2		2018 RW19	0.037	920	201	23.00	High	--
13	0.64	269.1	0.64	265.6		2000 EE117	0.020	271	107	19.80	High	--
14	0.19	236.0	---	---	---	---	---	921	587	14.63	None	--
15	0.48	245.9	0.50	249.4		2005 EQ301	0.065	999	380	14.46	High	--
16	0.13	232.9	0.51	246.2		2016 CM211	1.046	1370	807	11.93	None	--
17	0.52	237.3	---	---	---	---	---	1346	238	11.62	High	--
18	0.54	238.9	---	---	---	---	---	1453	199	11.49	None	--
19	3.39	332.8	---	---	---	---	---	873	747	10.75	None	--
20	0.25	309.5	---	---	---	---	---	182	522	10.69	None	--

Figure 51 - 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.

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 (sometimes more). 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 *Tools->Calculate Optimal Exposure Time* from the main menu. This window has its own "Help" file, which you can access by clicking on the "Help…" button.

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, alignment, and plate solving. 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 a machine equipped with an NVIDIA RTX 2080 GPU, it took around 348 seconds to complete.

Step 3: Analyze the Tracks

After the tracker has finished, you will note that it has found a total of 17 asteroids: the first 16 are rated "high" confidence, then another "Med" (medium) confidence at track #21. Your track numbers may be somewhat different if the images were processed with a different alignment option. 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, alignment, and plate solving.

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).

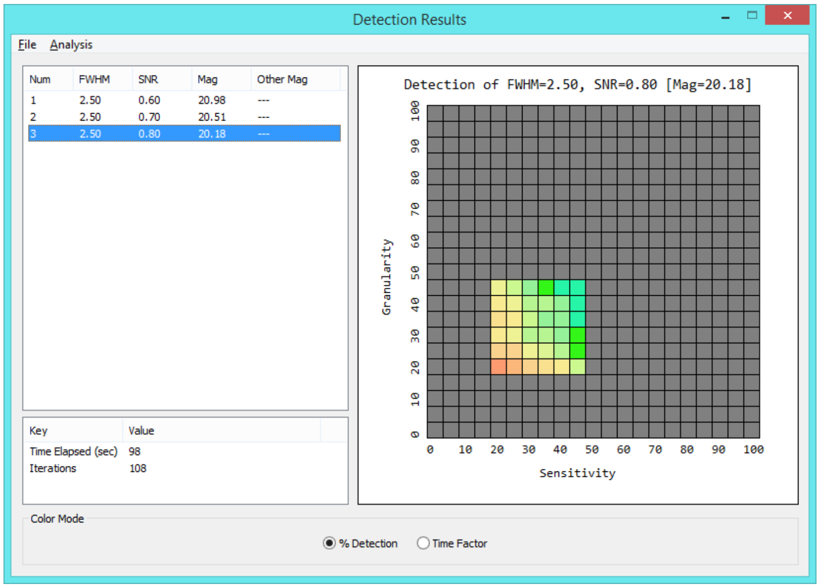


Figure 52 - Detection with 39 Images

Step 4: Compare with only 13 Images

The previous step demonstrated that it is possible to detect 92 out of the 100 injected targets at SNR=0.8, which corresponds to approximately magnitude 20.2 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 53:

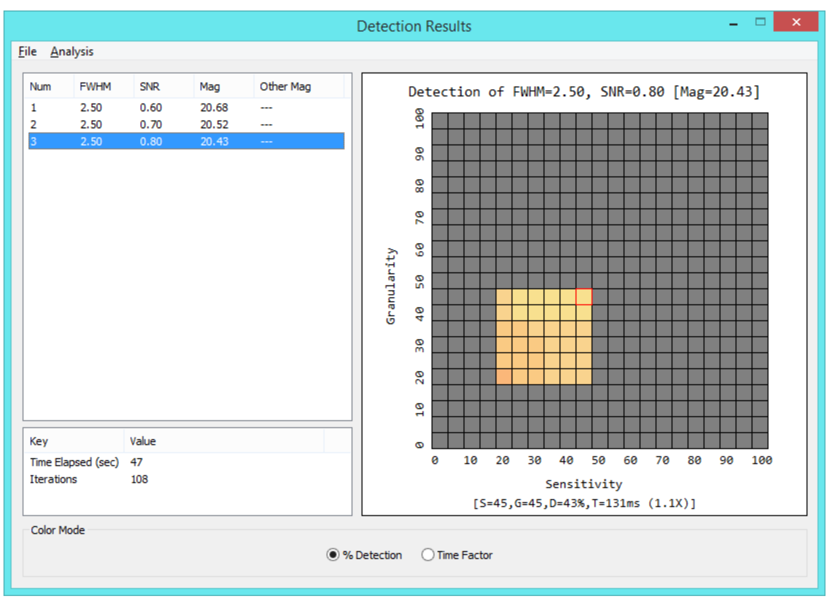


Figure 53 - Detection with Only 13 Images

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.

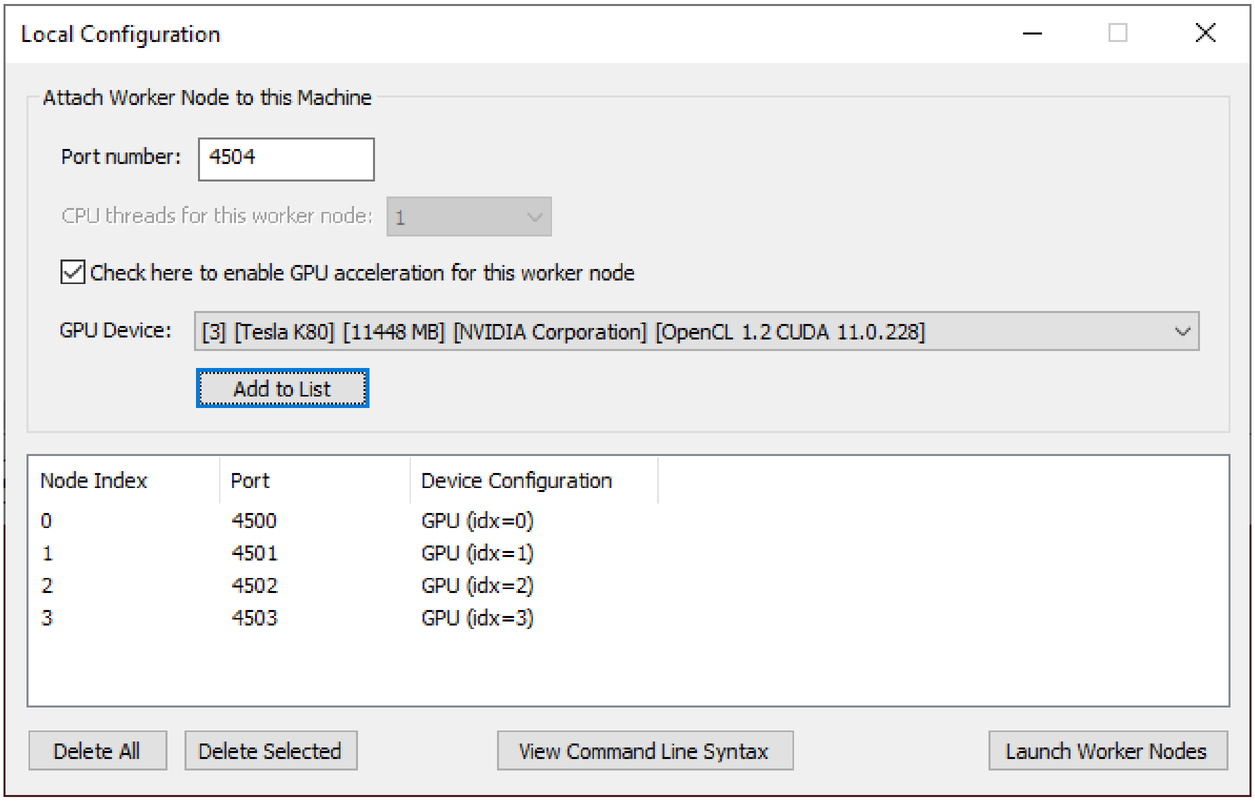


Figure 54 – Local Worker Node Configuration

Figure 54 shows the local configuration for a single machine that has four GPUs. 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 55.

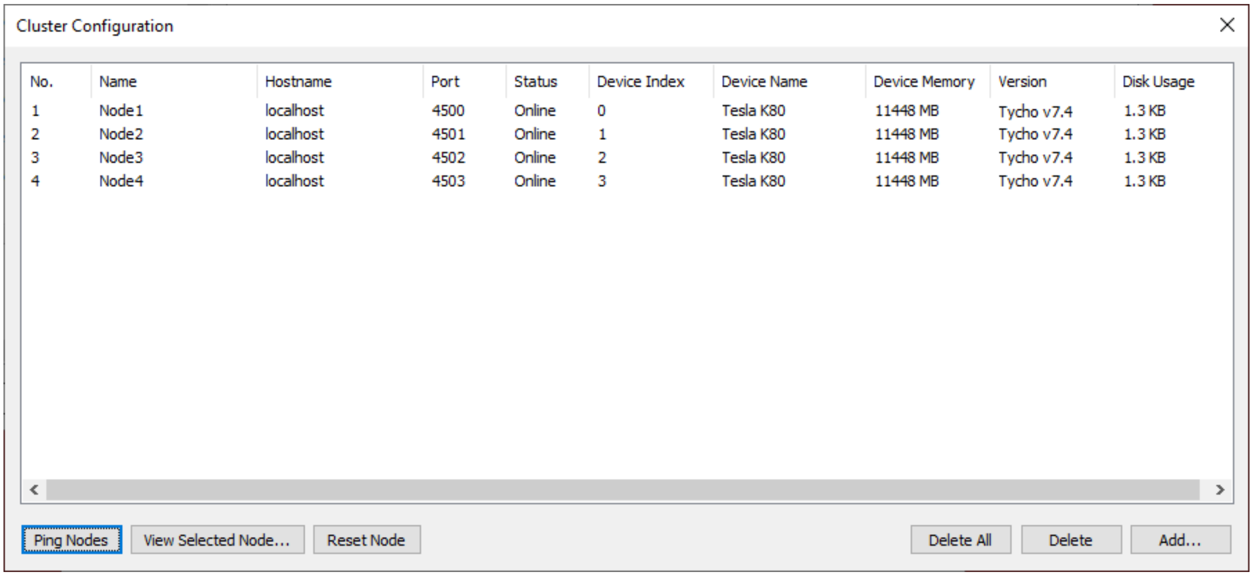


Figure 55 - Managing Worker Nodes

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 55 you can see that four nodes were added, one for each of the worker nodes configured in the previous step. You can 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.

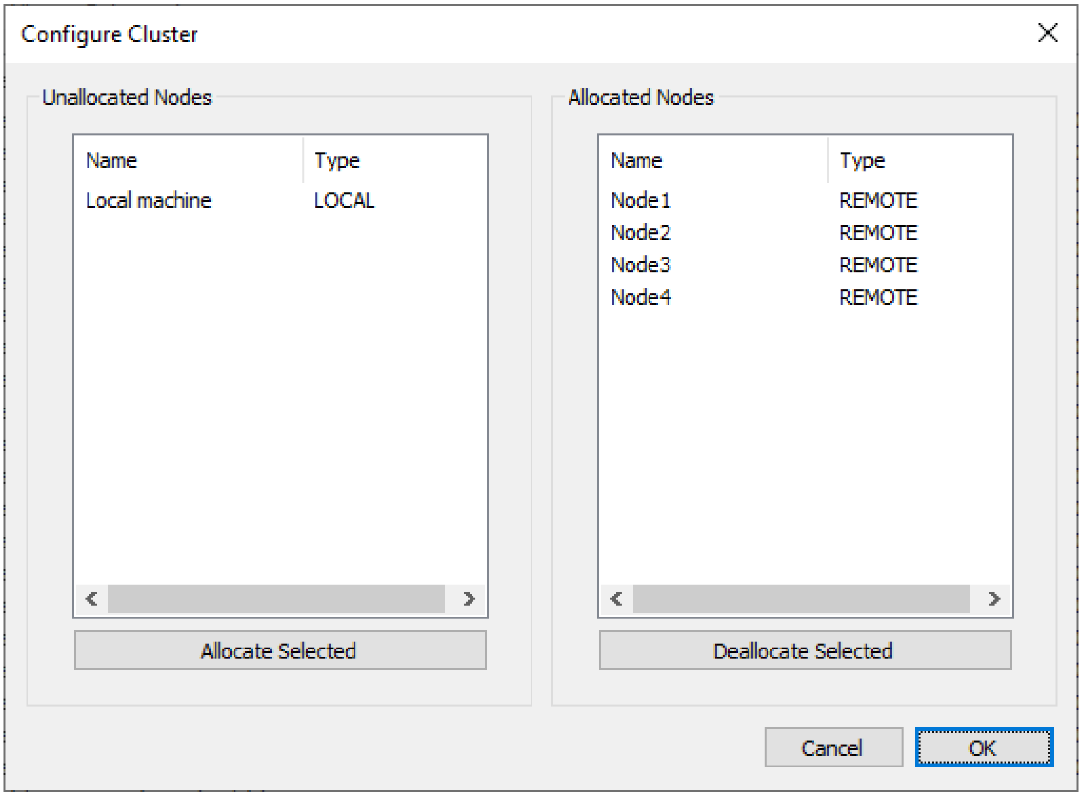


Figure 56 - Cluster Configuration

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 57. 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.

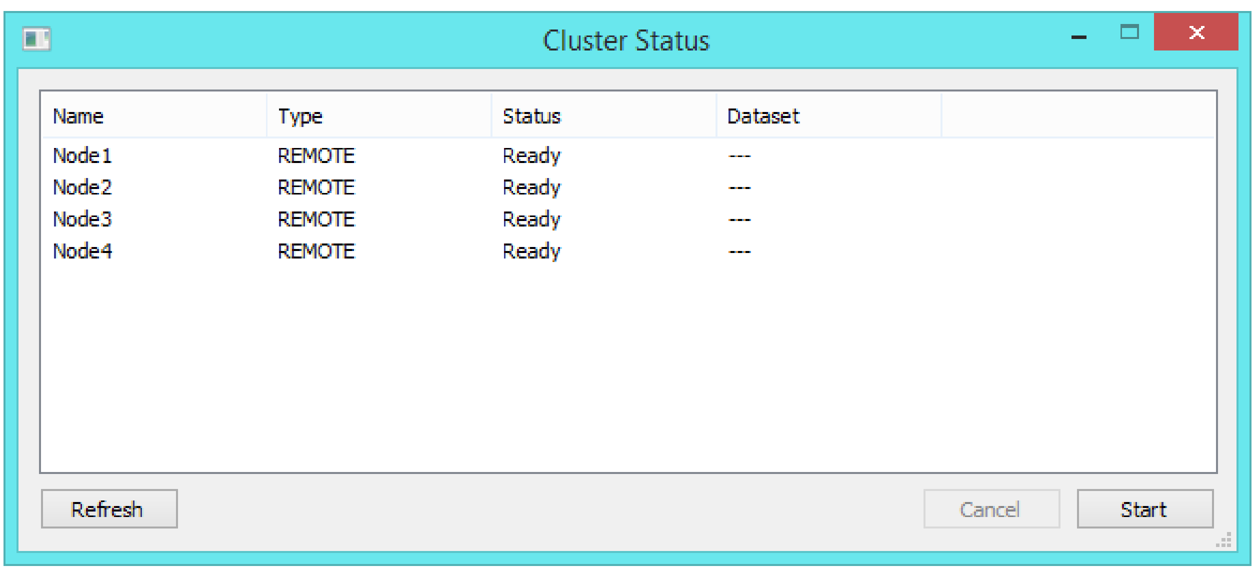


Figure 57 - Cluster Status

As each node processes the data, you will see the status of each node shown in the "Status" column. Refer to Figure 58.

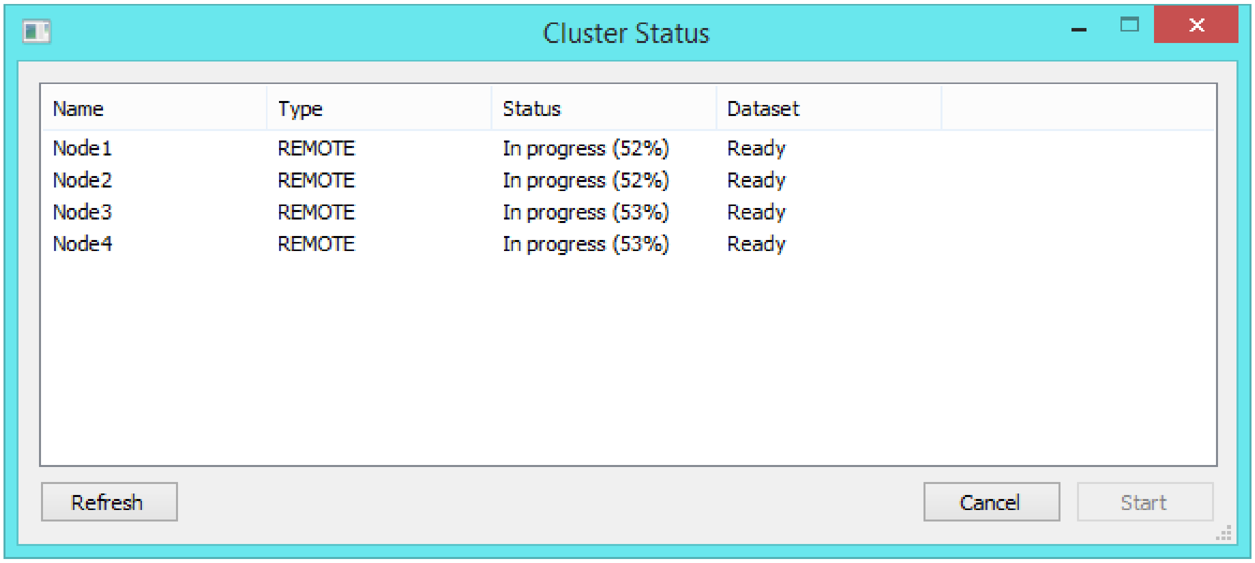


Figure 58 - Node Progress

In this example, because each node is running an equivalent GPU, they each have almost identical progress. Note that there are some circumstances in which it does not make sense to use multiple GPUs, such as when one GPU is more than twice as fast as another. In that case, the other GPU would simply cause the overall processing time to increase since the workload is distributed evenly between the fast and slow GPU. In general, no single node should be more than N times as fast as any other node, where N is the number of nodes.

Express Mode

Tycho can also combine the processing steps (calibration, alignment, and plate solving) 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 59 for details.

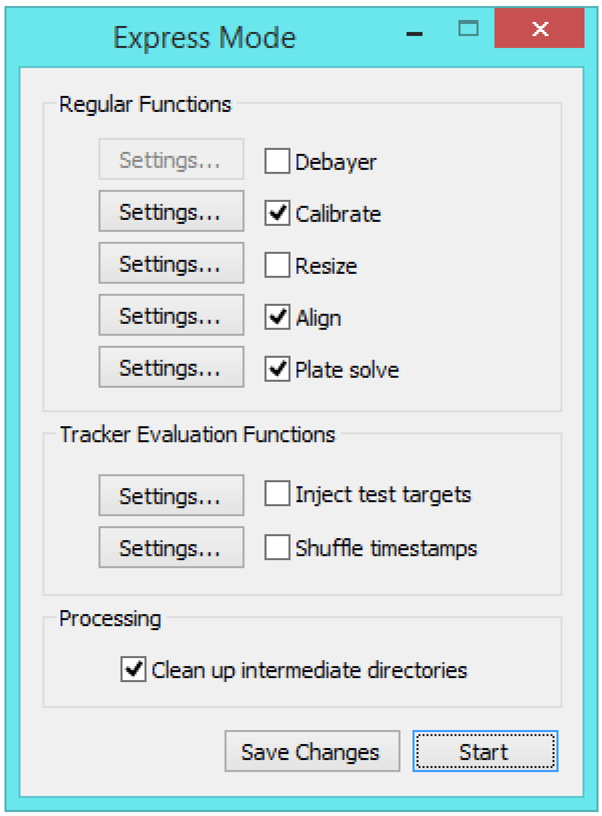


Figure 59 - Express Mode

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.

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 60. 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.

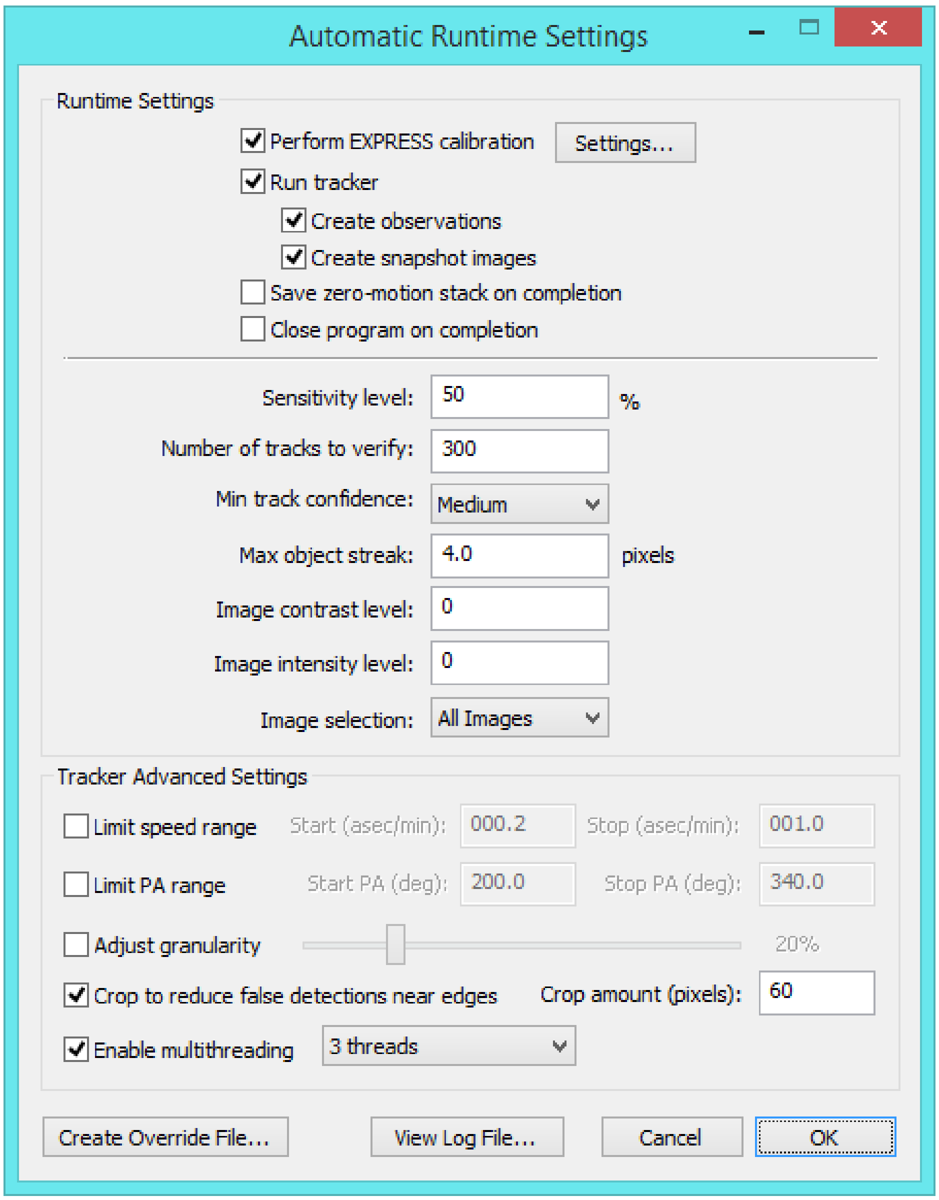


Figure 60 - Auto Run Settings

Once you have configured the desired settings for Auto Run, click “OK” to save the settings. Then, you can open a command 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:
"C:\Program Files\Tycho\Tycho.exe" 1 "C:\Users\Daniel\Desktop\iTelescope\data\pipeline\r1"

The above command will launch Tycho in auto run mode and process the files located in the ‘r1’ subdirectory.

Another way to use the Auto Run feature is to create a batch file, such as “auto_run.bat”, 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 60. 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.

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).

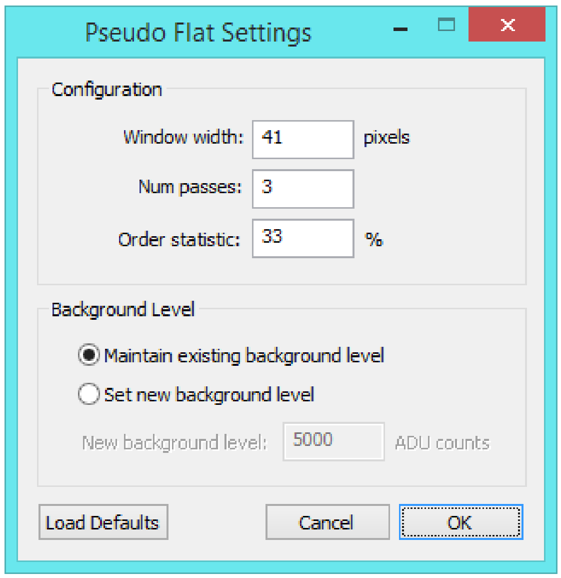


Figure 61 - Image Calibration

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 for 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:

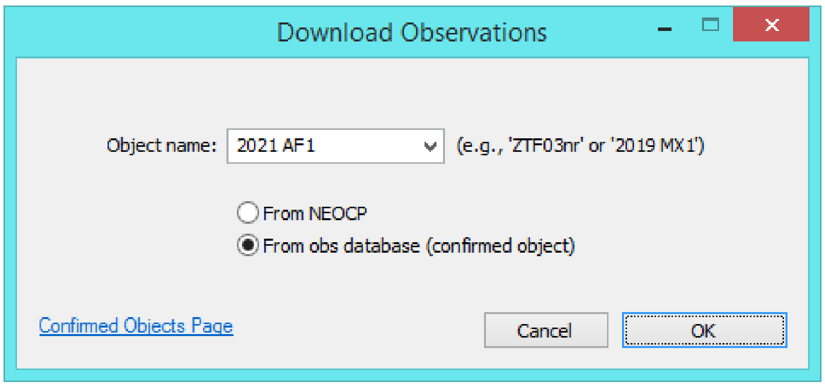


Figure 62 - Download Observations

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).

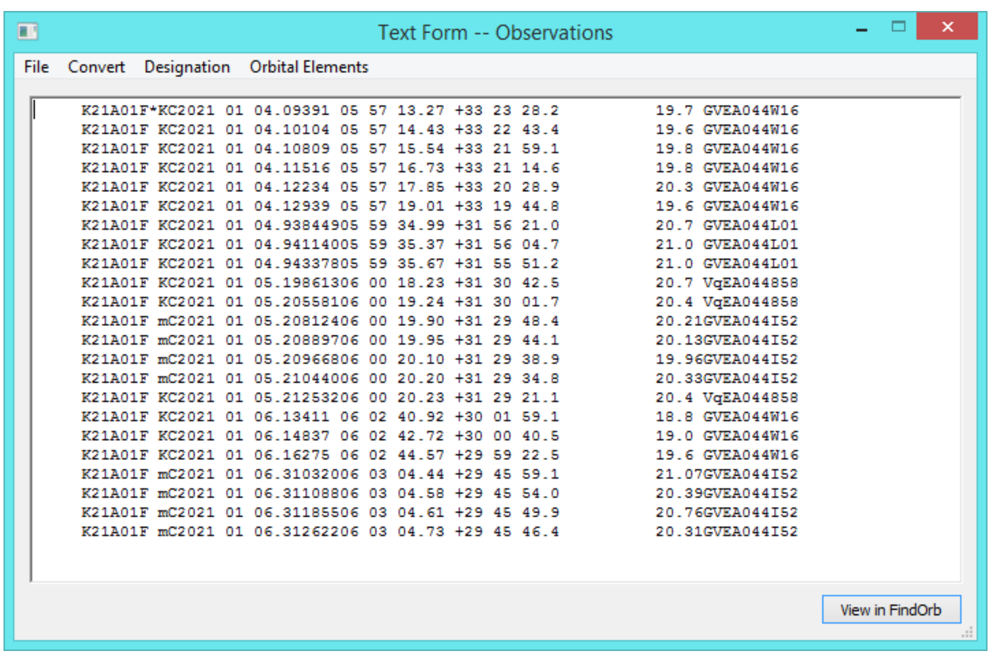


Figure 63 - Downloaded Observations

At this point, you can click “View in Find_Orb”, and the modified version of *Find_Orb* (find_o64_modified.exe) will be invoked. If you do not have this modified version, please refer to the downloads section on the Tycho website. Next, while keeping *Find_Orb* running in the background, navigate back to the “Image Manager” and choose *Ephemeris->Attach to Dataset*. The ephemeris information of the object is now 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 “EPH_IN_FOV” column, 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.

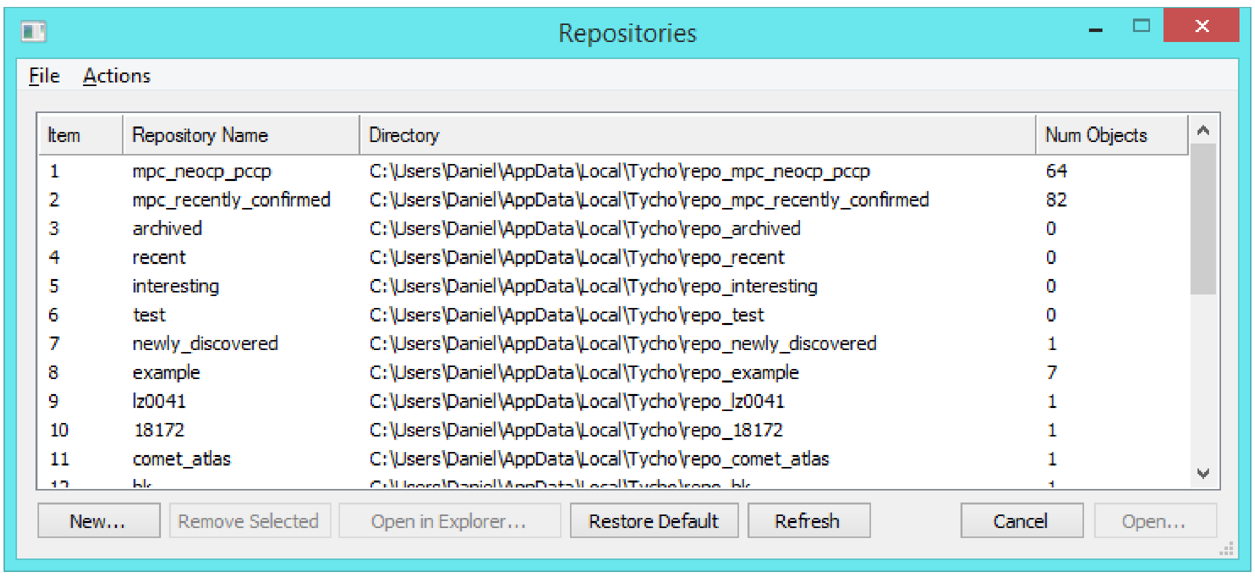


Figure 64 - Manage Repositories

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 "Add…" from the Repositories window.

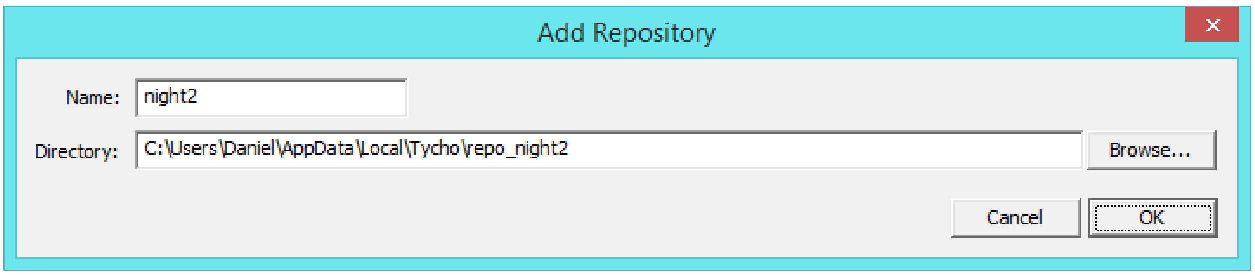


Figure 65 - Adding a Repository

Removing a Repository

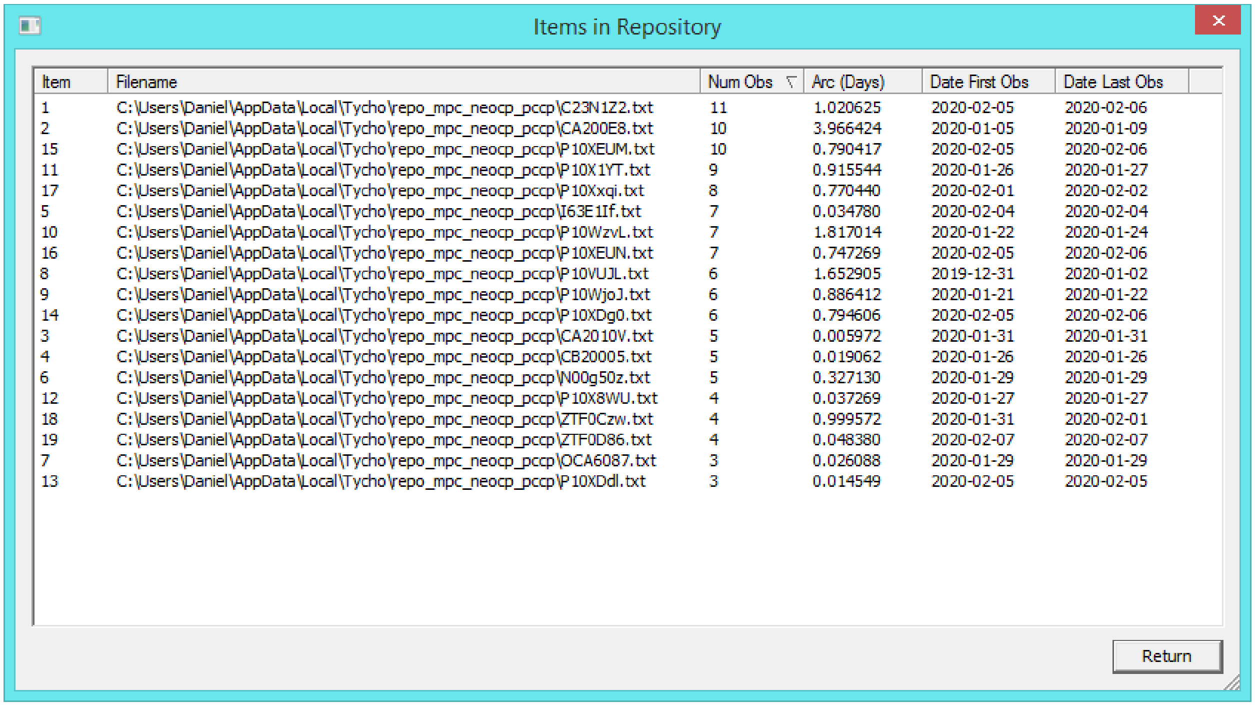
A repository can be removed from the list of repositories by selecting it from the list and clicking the "Remove Selected" button from the Repositories window. Alternatively, one could also right-click the repository and choose "Remove from List". Note that removing a repository simply removes it from 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" 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 external to the Repository Manager (such as via Explorer), 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 simply choose "Open in Explorer" 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 in Find_Orb". The first option, "View", will copy the observations of the object to the text window ("Text Form – Observations"). The latter option, "View in Find_Orb", shows the observations in Find_Orb.



Item	Filename	Num Obs	Arc (Days)	Date First Obs	Date Last Obs
1	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\C23N1Z2.txt	11	1.020625	2020-02-05	2020-02-06
2	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\CA200E8.txt	10	3.966424	2020-01-05	2020-01-09
15	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\P10XEUM.txt	10	0.790417	2020-02-05	2020-02-06
11	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\P10X1YT.txt	9	0.915544	2020-01-26	2020-01-27
17	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\P10Xxqi.txt	8	0.770440	2020-02-01	2020-02-02
5	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\I63E1If.txt	7	0.034780	2020-02-04	2020-02-04
10	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\P10WzvL.txt	7	1.817014	2020-01-22	2020-01-24
16	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\P10XEUN.txt	7	0.747269	2020-02-05	2020-02-06
8	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\P10VUJL.txt	6	1.652905	2019-12-31	2020-01-02
9	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\P10WjoJ.txt	6	0.886412	2020-01-21	2020-01-22
14	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\P10XDg0.txt	6	0.794606	2020-02-05	2020-02-06
3	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\CA2010V.txt	5	0.005972	2020-01-31	2020-01-31
4	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\CB20005.txt	5	0.019062	2020-01-26	2020-01-26
6	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\N00g50z.txt	5	0.327130	2020-01-29	2020-01-29
12	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\P10X8WU.txt	4	0.037269	2020-01-27	2020-01-27
18	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\ZTF0Czw.txt	4	0.999572	2020-01-31	2020-02-01
19	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\ZTF0D86.txt	4	0.048380	2020-02-07	2020-02-07
7	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\OCA6087.txt	3	0.026088	2020-01-29	2020-01-29
13	C:\Users\Daniel\AppData\Local\Tycho\repo_mpc_neocp_pccp\P10XDdl.txt	3	0.014549	2020-02-05	2020-02-05

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 new 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 "P10XEUM". 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.

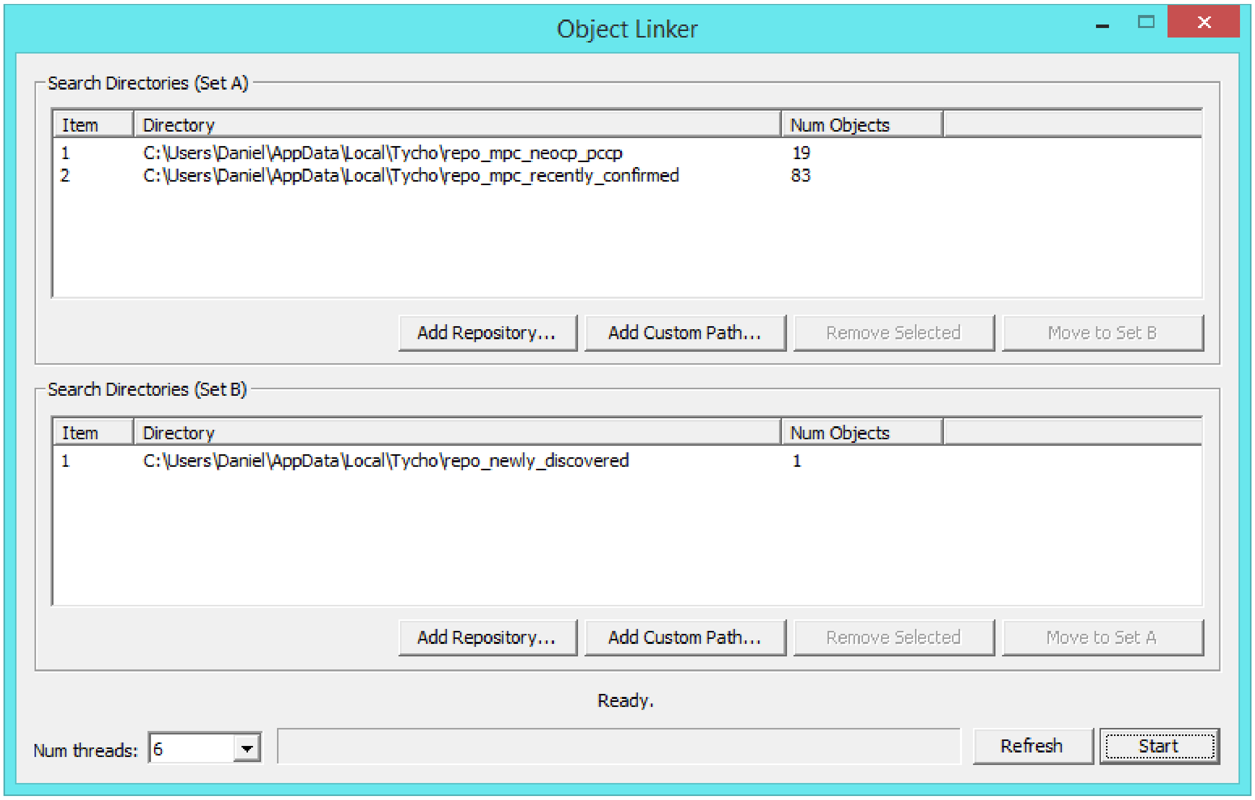


Figure 67 - Object Linker

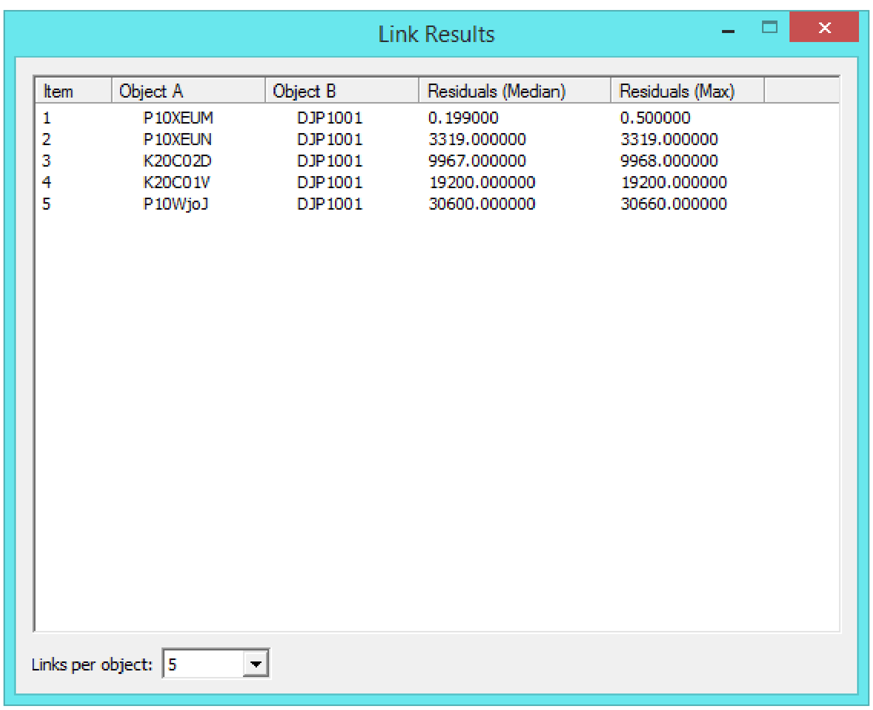


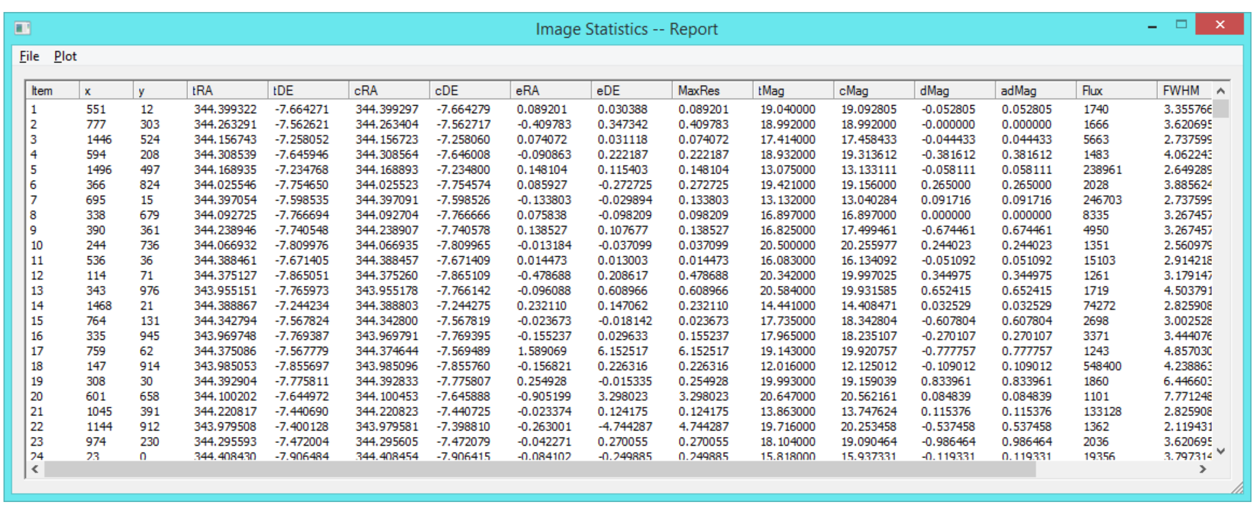
Figure 68 - Link Results

Continuing with this example, you would first proceed to 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 for 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 at random 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).



Item	x	y	tRA	tDE	cRA	cDE	eRA	eDE	MaxRes	tMag	cMag	dMag	adMag	Flux	FWHM
1	551	12	344.399322	-7.664271	344.399297	-7.664279	0.089201	0.030388	0.089201	19.040000	19.092805	-0.052805	0.052805	1740	3.355766
2	777	303	344.263291	-7.562621	344.263404	-7.562717	-0.409783	0.347342	0.409783	18.992000	18.992000	-0.000000	0.000000	1666	3.620695
3	1446	524	344.156743	-7.258052	344.156723	-7.258060	0.074072	0.031118	0.074072	17.414000	17.458433	-0.044433	0.044433	5663	2.737599
4	594	208	344.308539	-7.645946	344.308564	-7.646008	-0.090863	0.222187	0.222187	18.932000	19.313612	-0.381612	0.381612	1483	4.062243
5	1496	497	344.168935	-7.234768	344.168893	-7.234800	0.148104	0.115403	0.148104	13.075000	13.133111	-0.058111	0.058111	238961	2.649289
6	366	824	344.025546	-7.754650	344.025523	-7.754574	0.085927	-0.272725	0.272725	19.421000	19.156000	0.265000	0.265000	2028	3.885624
7	695	15	344.397054	-7.598535	344.397091	-7.598526	-0.133803	-0.029894	0.133803	13.132000	13.040284	0.091716	0.091716	246703	2.737599
8	338	679	344.092725	-7.766694	344.092704	-7.766666	0.075838	-0.098209	0.098209	16.897000	16.897000	0.000000	0.000000	8335	3.267457
9	390	361	344.238946	-7.740548	344.238907	-7.740578	0.138527	0.107677	0.138527	16.825000	17.499461	-0.674461	0.674461	4950	3.267457
10	244	736	344.066932	-7.809976	344.066935	-7.809965	-0.013184	-0.037099	0.037099	20.500000	20.255977	0.244023	0.244023	1351	2.560979
11	536	36	344.388461	-7.671405	344.388457	-7.671409	0.014473	0.013003	0.014473	16.083000	16.134092	-0.051092	0.051092	15103	2.914218
12	114	71	344.375127	-7.865051	344.375260	-7.865109	-0.478688	0.208617	0.478688	20.342000	19.997025	0.344975	0.344975	1261	3.179147
13	343	976	343.955151	-7.765973	343.955178	-7.766142	-0.096088	0.608966	0.608966	20.584000	19.931585	0.652415	0.652415	1719	4.503791
14	1468	21	344.388867	-7.244234	344.388803	-7.244275	0.232110	0.147062	0.232110	14.441000	14.408471	0.032529	0.032529	74272	2.825908
15	764	131	344.342794	-7.567824	344.342800	-7.567819	-0.023673	-0.018142	0.023673	17.735000	18.342804	-0.607804	0.607804	2698	3.002528
16	335	945	343.969748	-7.769387	343.969791	-7.769395	-0.155237	0.029633	0.155237	17.965000	18.235107	-0.270107	0.270107	3371	3.444076
17	759	62	344.375086	-7.567779	344.374644	-7.569489	1.589069	6.152517	6.152517	19.143000	19.920757	-0.777757	0.777757	1243	4.857030
18	147	914	343.985053	-7.855697	343.985096	-7.855760	-0.156821	0.226316	0.226316	12.016000	12.125012	-0.109012	0.109012	548400	4.238863
19	308	30	344.392904	-7.775811	344.392833	-7.775807	0.254928	-0.015335	0.254928	19.993000	19.159039	0.833961	0.833961	1860	6.446603
20	601	658	344.100202	-7.644972	344.100453	-7.645888	-0.905199	3.298023	3.298023	20.647000	20.562161	0.084839	0.084839	1101	7.771248
21	1045	391	344.220817	-7.440690	344.220823	-7.440725	-0.023374	0.124175	0.124175	13.863000	13.747624	0.115376	0.115376	133128	2.825908
22	1144	912	343.979508	-7.400128	343.979581	-7.398810	-0.263001	-4.744287	4.744287	19.716000	20.253458	-0.537458	0.537458	1362	2.119431
23	974	230	344.295593	-7.472004	344.295605	-7.472079	-0.042271	0.270055	0.270055	18.104000	19.090464	-0.986464	0.986464	2036	3.620695
24	23	0	344.408430	-7.906484	344.408454	-7.906415	-0.084102	-0.249885	0.249885	15.818000	15.937331	-0.119331	0.119331	19356	3.797314

Figure 69 - Image Statistics

A number of fields 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.

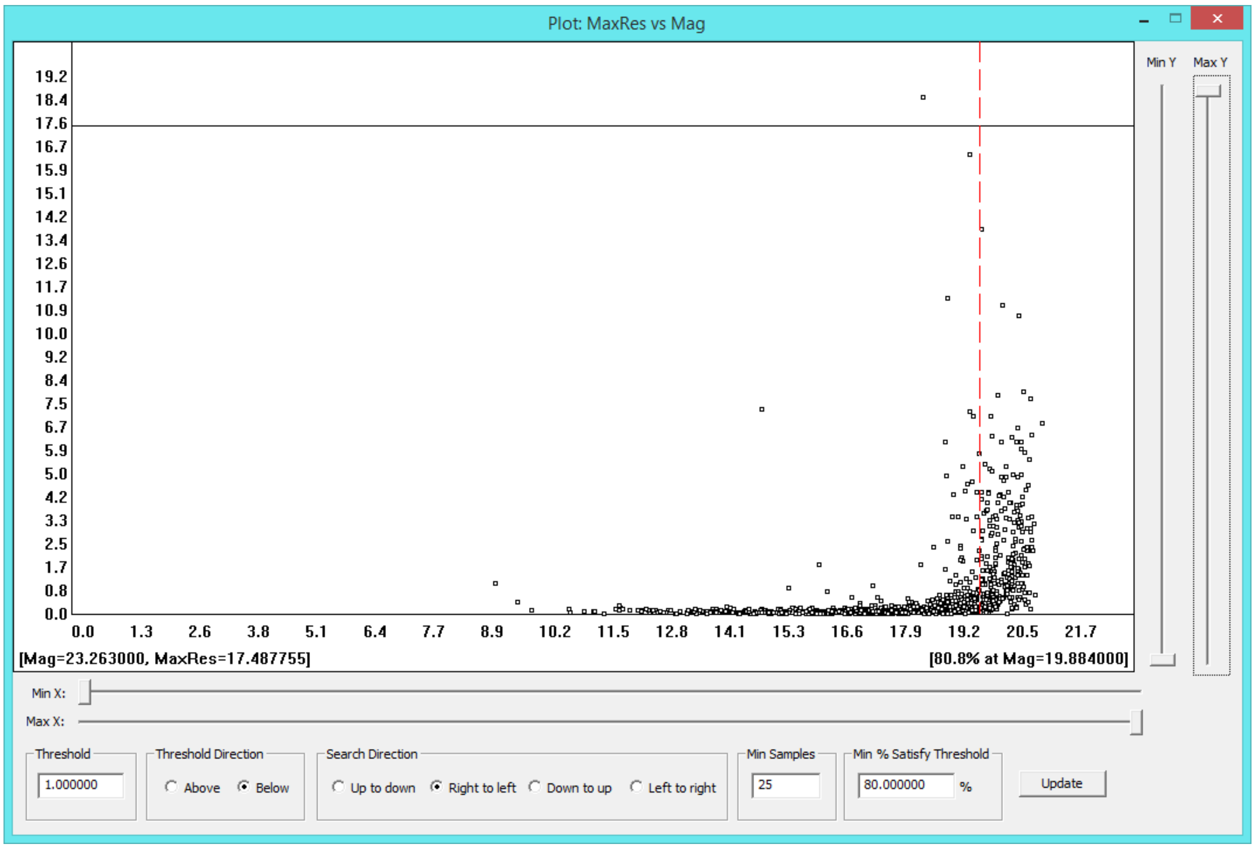


Figure 70 - Plot of MaxRes vs Magnitude

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, which 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.

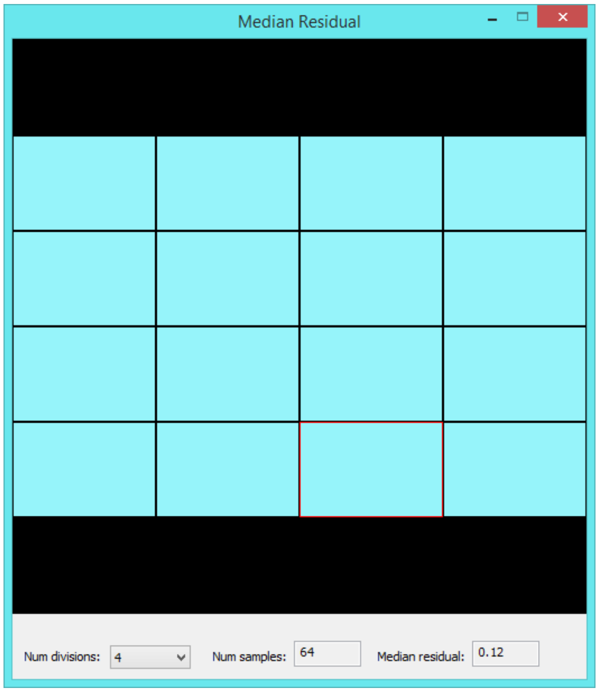


Figure 71 - Example of Dataset with Excellent Residuals

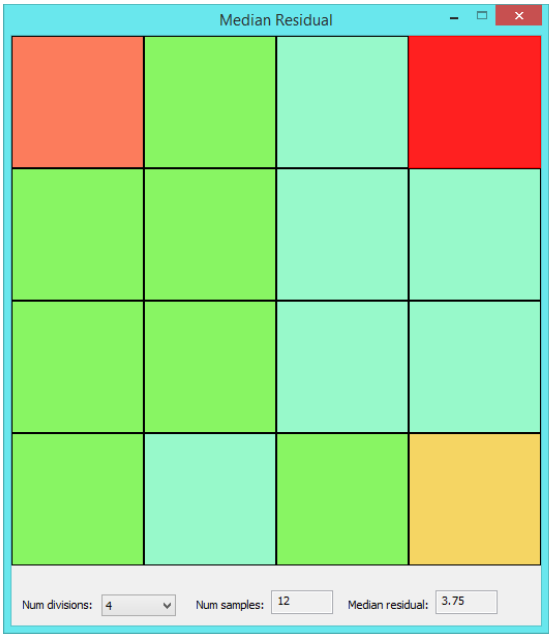


Figure 72 - Example of Dataset with Poor Residuals

In Figure 72, the grid indicates that the dataset tends to have acceptable residuals in the inner portion of the image, but poor residuals at the corners. When inspecting the images of this dataset, the stars at the corners are shown to have very soft focus, while the stars at the inner region have sharp focus. This matches well with the results shown in the grid. 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.

Constructing a Lightcurve

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

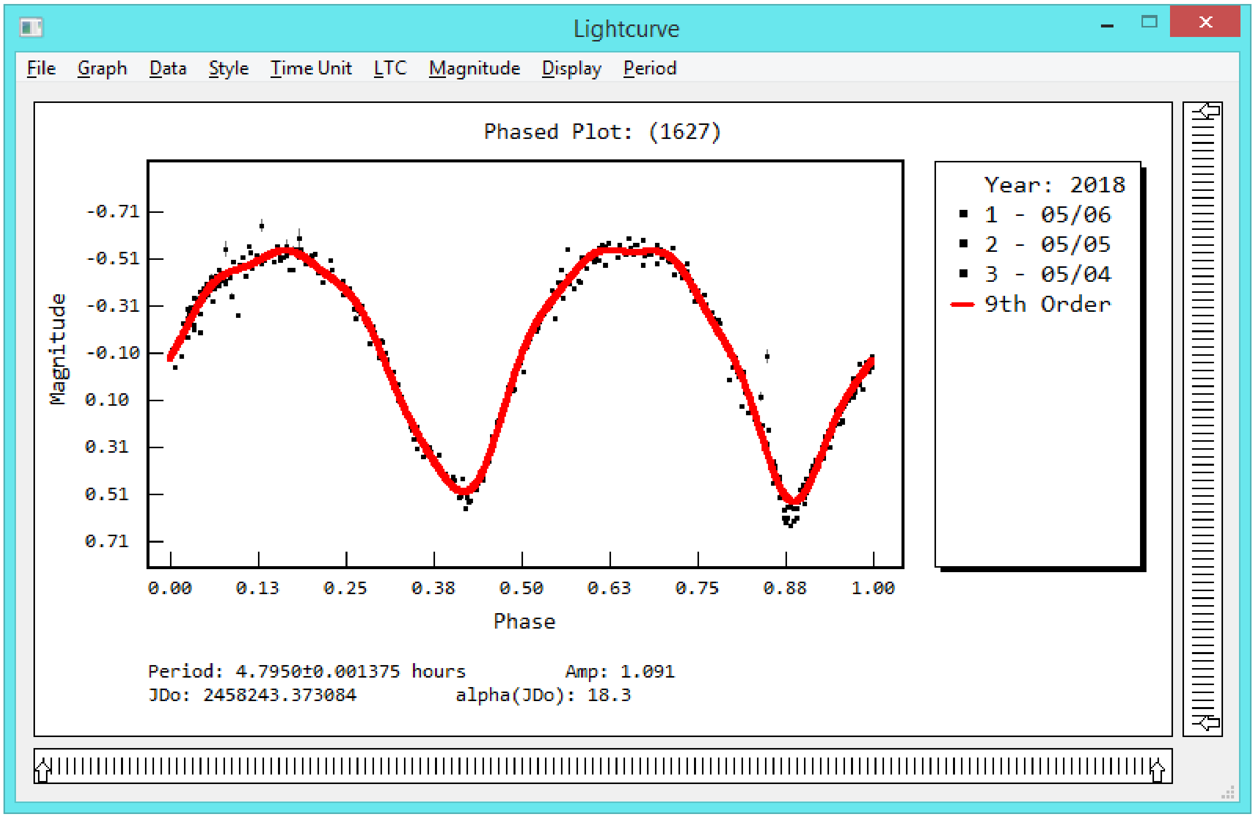


Figure 73 - Lightcurve of asteroid (1627) Ivar

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.

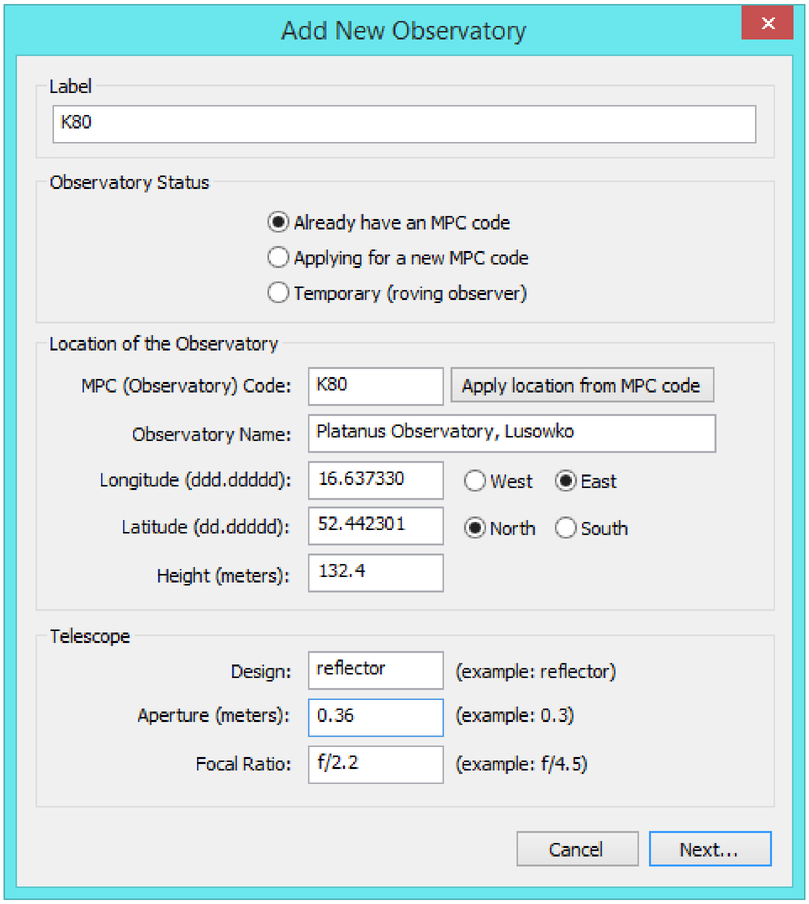


Figure 74 - Configuring Observatory 1/2

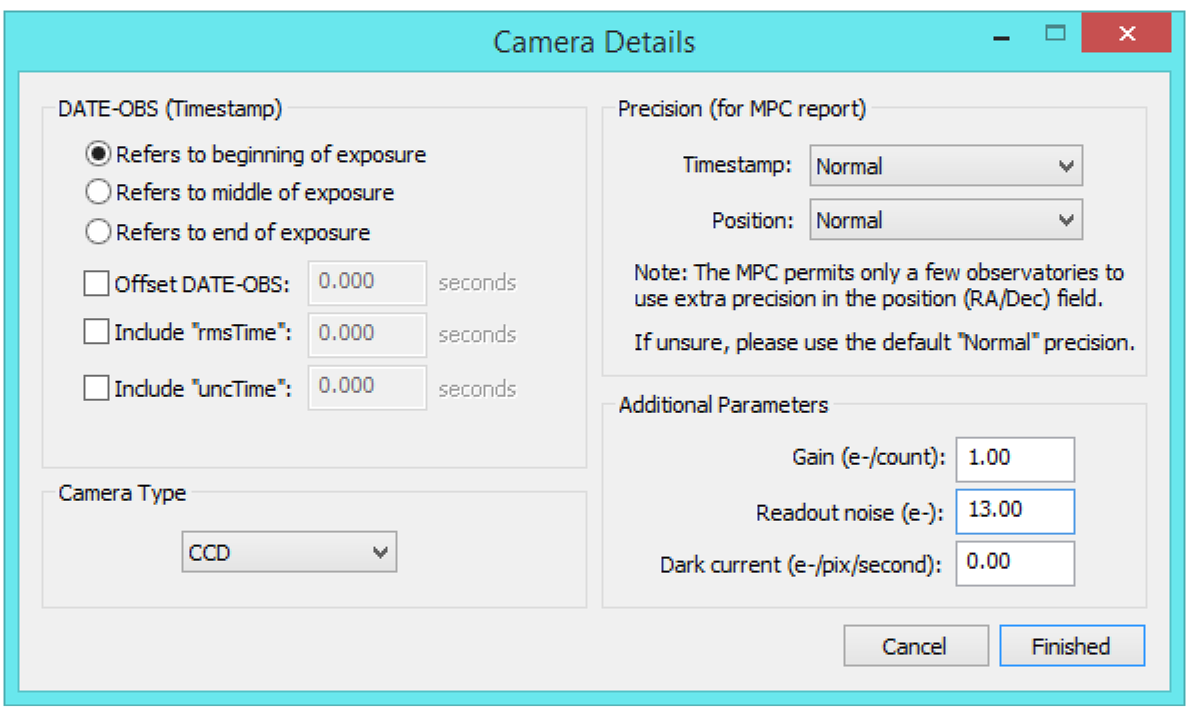


Figure 75 - Configuring Observatory 2/2

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.

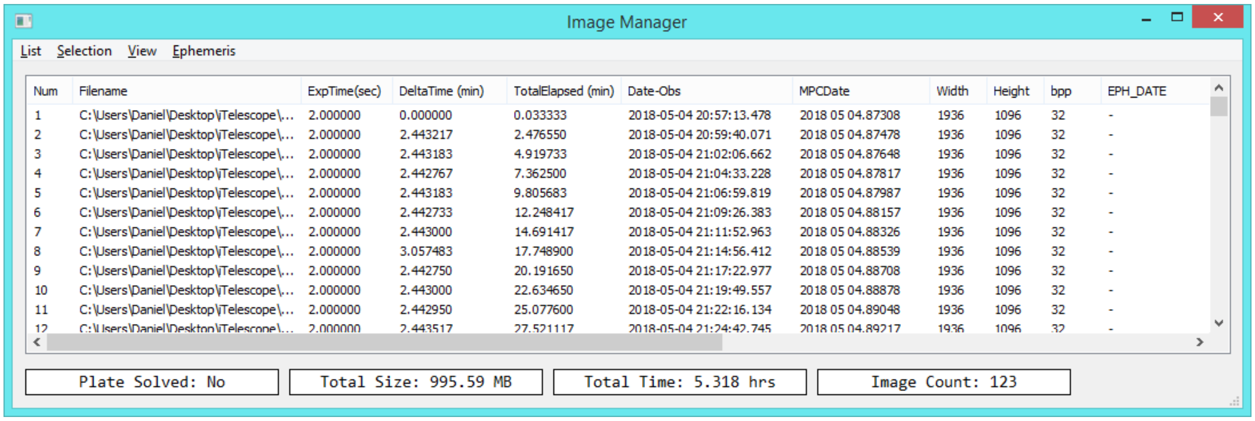


Figure 76 - The "Ivar_n1" Dataset with 123 Images

Step 3: Perform "Express Mode"

Navigate to *Action->Express Mode* and click the "Settings…" button located to the left of "Align". As there are a lot of images, you may want to choose the "Internal" option since it is very fast to process the 123 images. Then, navigate to the "Internal" tab to configure the settings. Ensure that the "Mode" is set to "Normal". Distortion correction is not required for these datasets (and would only increase the processing time), so leave it unchecked. Then click "Save Settings" to continue.

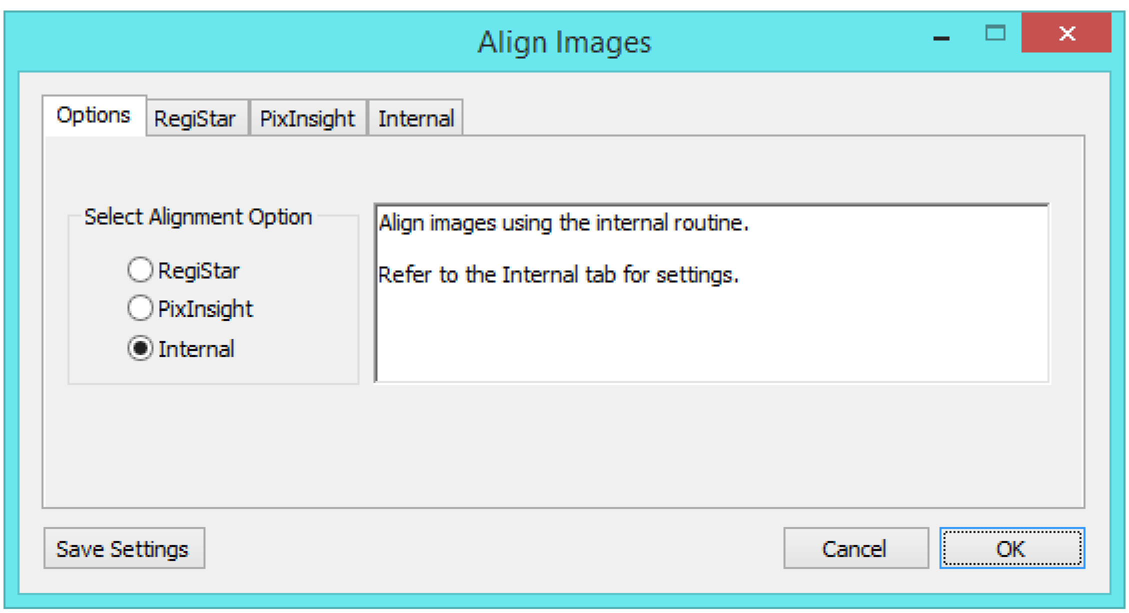


Figure 77 – Alignment Configuration

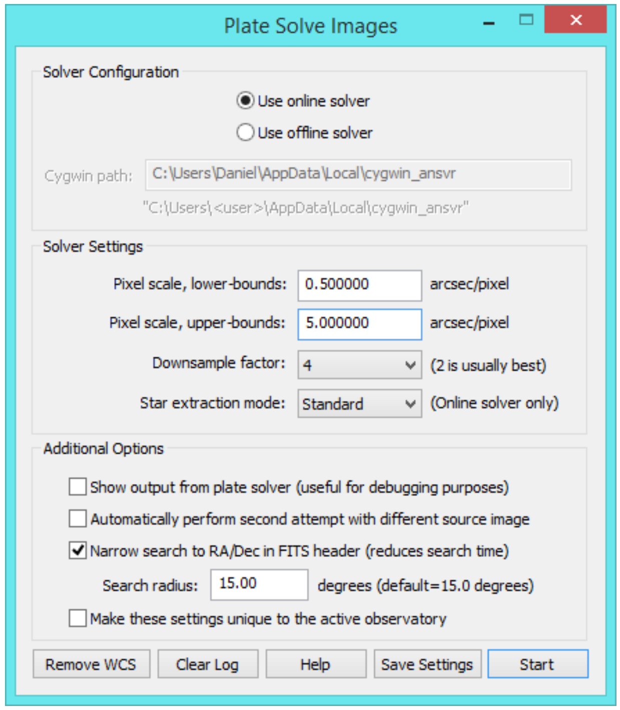


Figure 78 - Plate Solve Settings

Finally, back at the "Express Mode" window, 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.

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

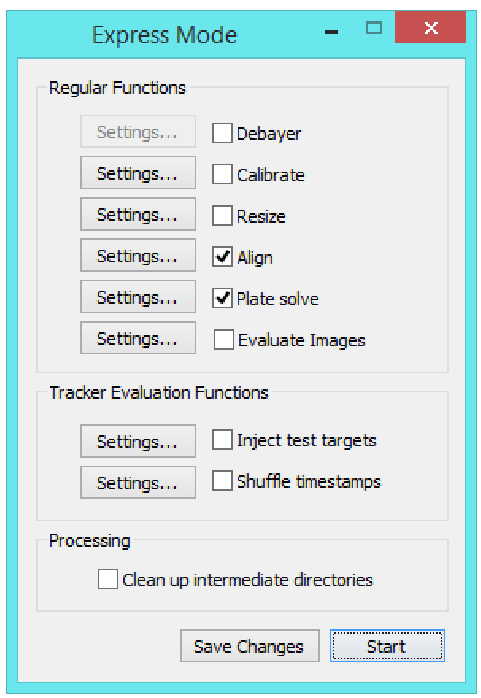


Figure 79 - Express Mode Settings

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 aligned and plate solved.

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".

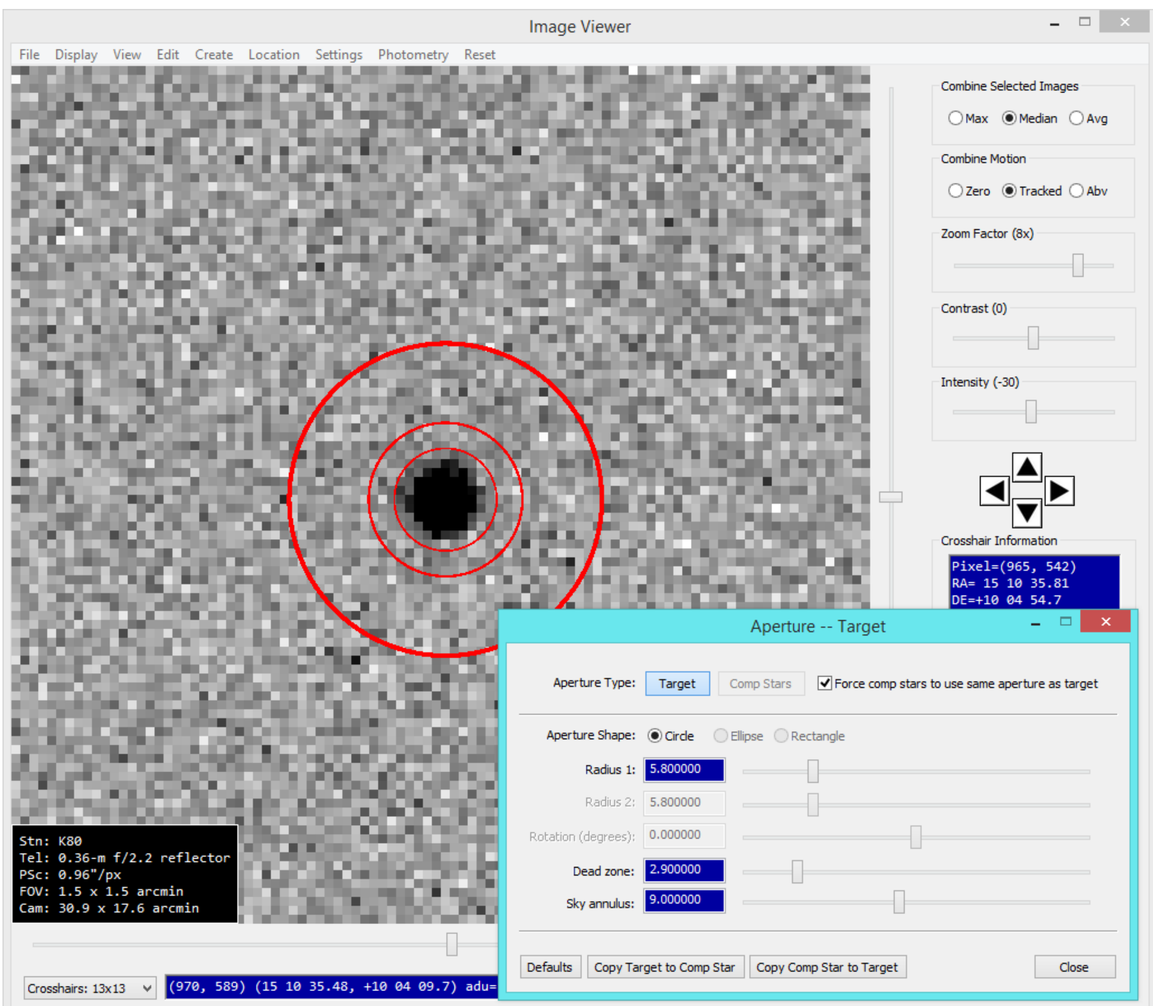


Figure 80 - Adjusting Aperture Settings

As shown in Figure 80, I have adjusted the radius 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.

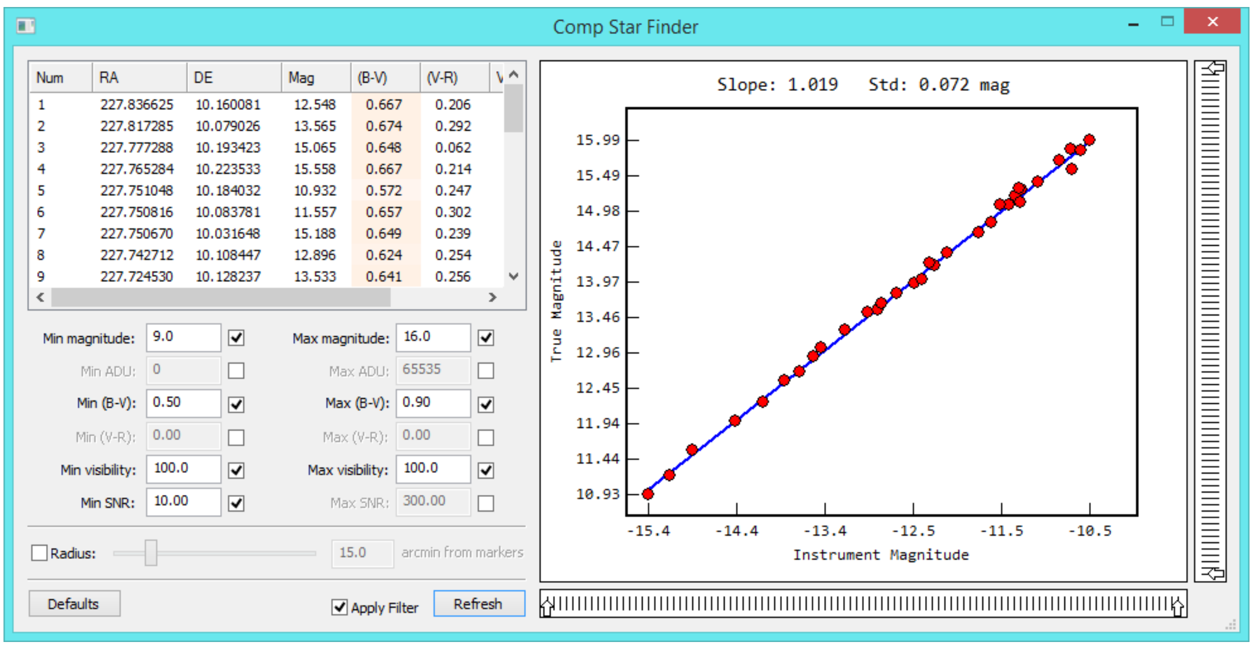


Figure 81 - Identify Optimal Comparison Stars

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 double-click on the star in the "Image Viewer", right-click, and choose "Add to Active Comp Stars" from that popup menu. 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.

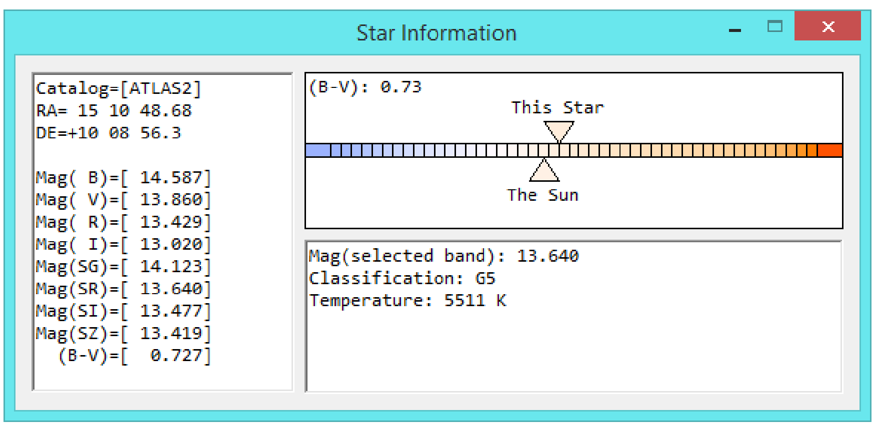


Figure 82 - View Star Information

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 Comp Star” from the popup menu that appears.

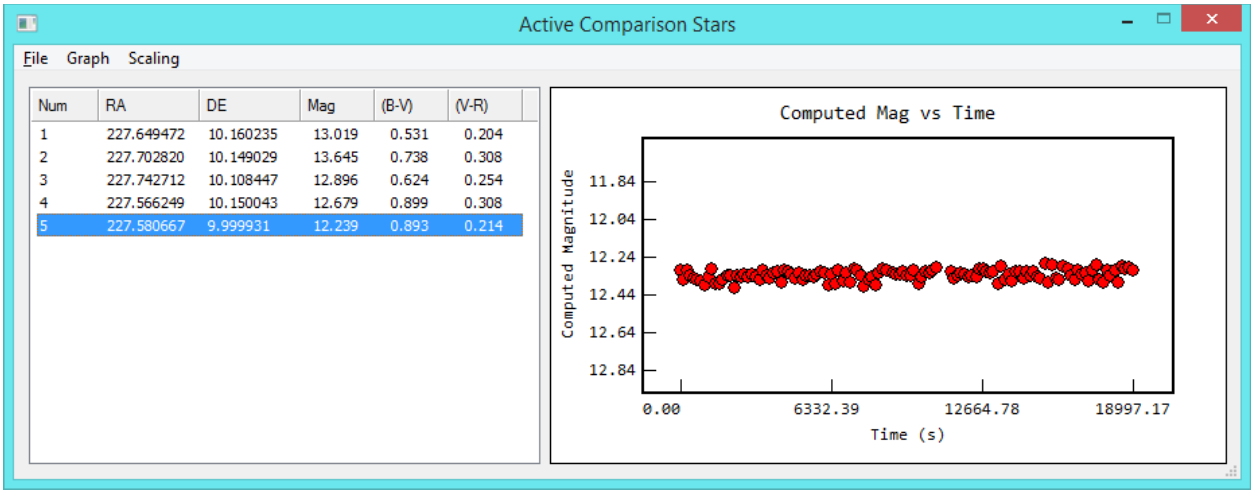


Figure 83 - Active Comparison Stars

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 Comp Star”. 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 (at 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 “Generate Photometry Set”. 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.

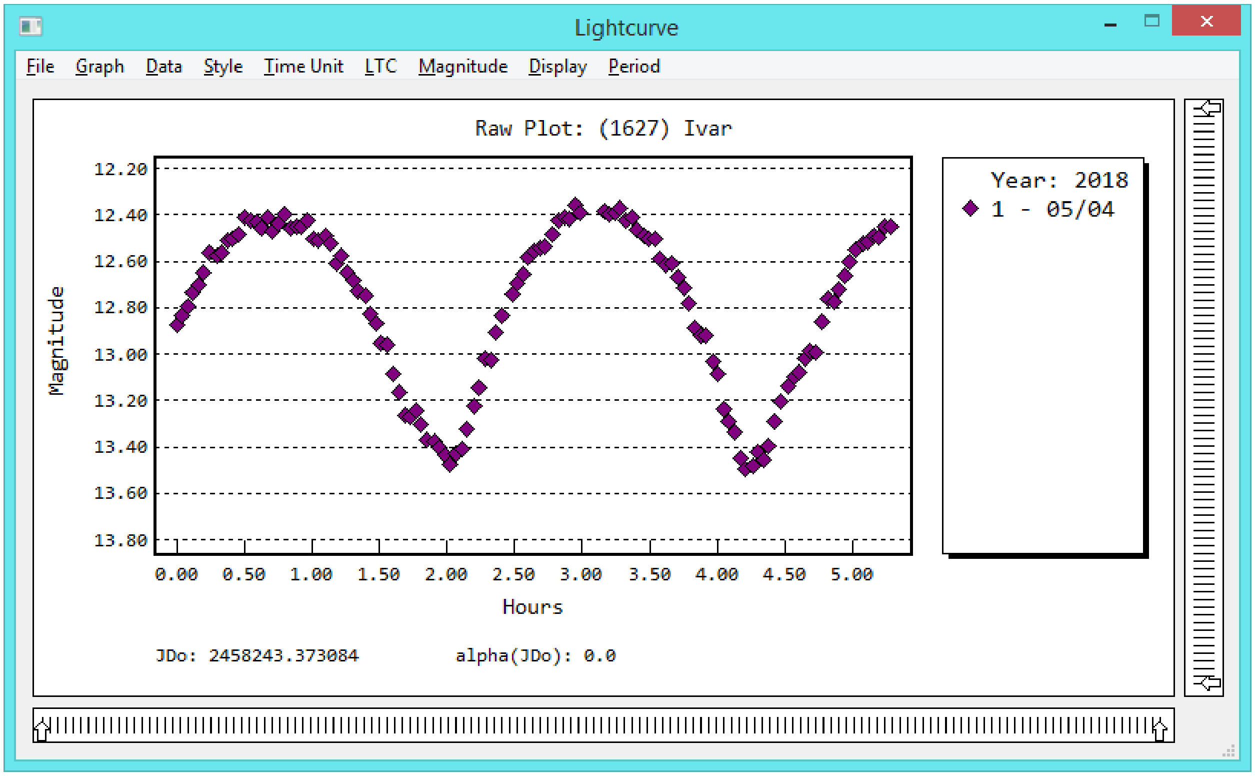


Figure 84 - First Night of Data for (1627) Ivar

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.

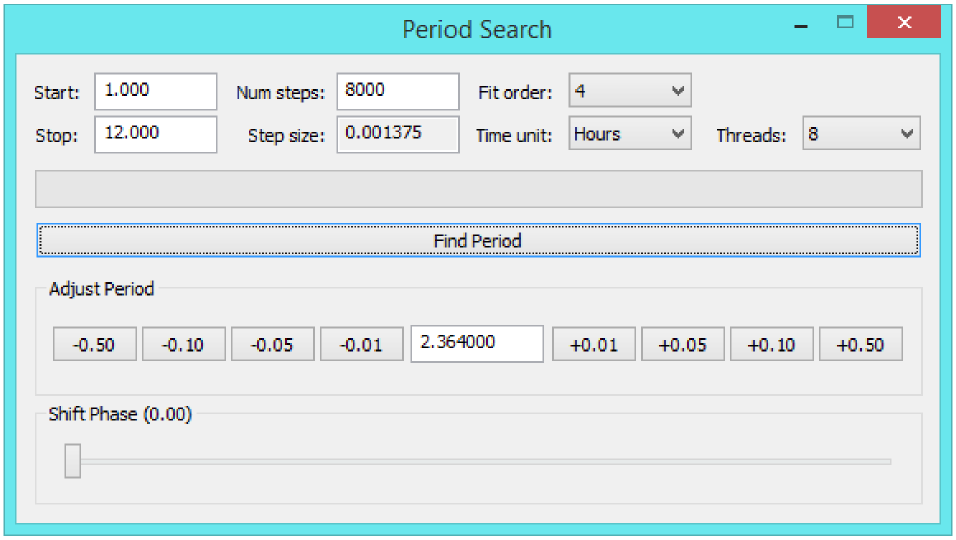
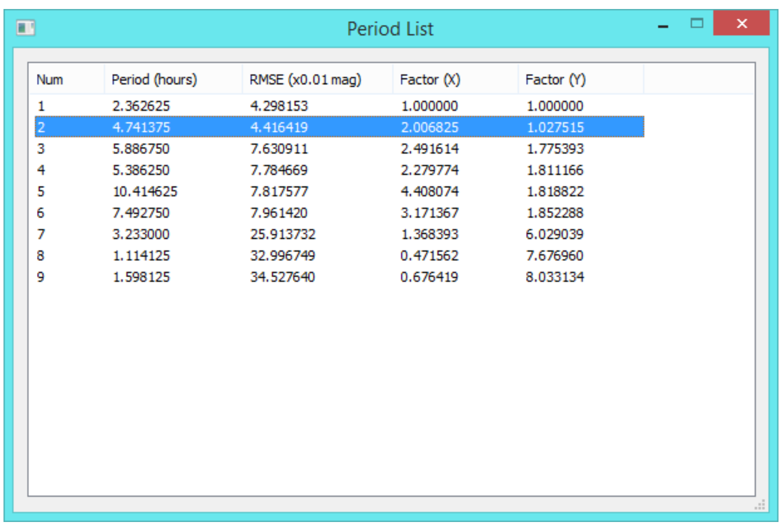


Figure 85 - Period Search Settings

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)	RMSE (x0.01 mag)	Factor (X)	Factor (Y)
1	2.362625	4.298153	1.000000	1.000000
2	4.741375	4.416419	2.006825	1.027515
3	5.886750	7.630911	2.491614	1.775393
4	5.386250	7.784669	2.279774	1.811166
5	10.414625	7.817577	4.408074	1.818822
6	7.492750	7.961420	3.171367	1.852288
7	3.233000	25.913732	1.368393	6.029039
8	1.114125	32.996749	0.471562	7.676960
9	1.598125	34.527640	0.676419	8.033134

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.

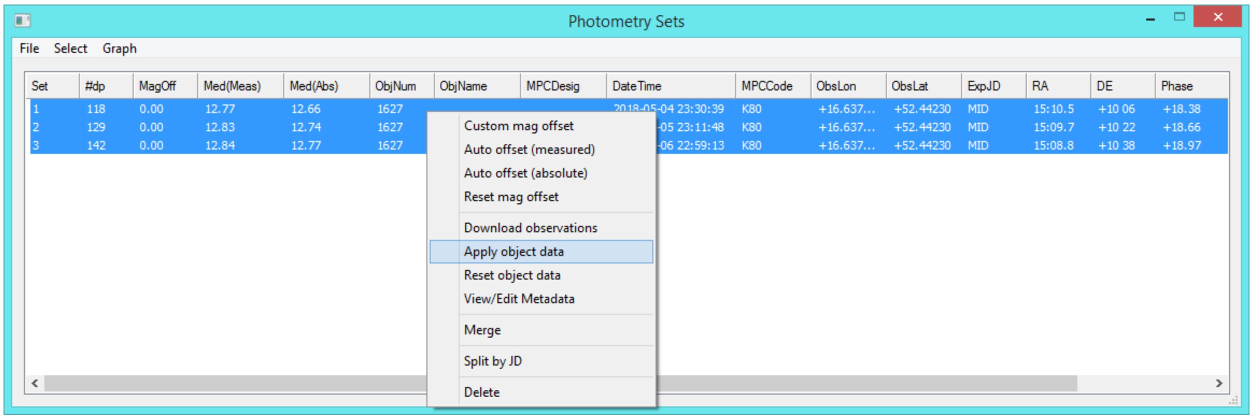


Figure 87 - Three Photometry Sets

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.

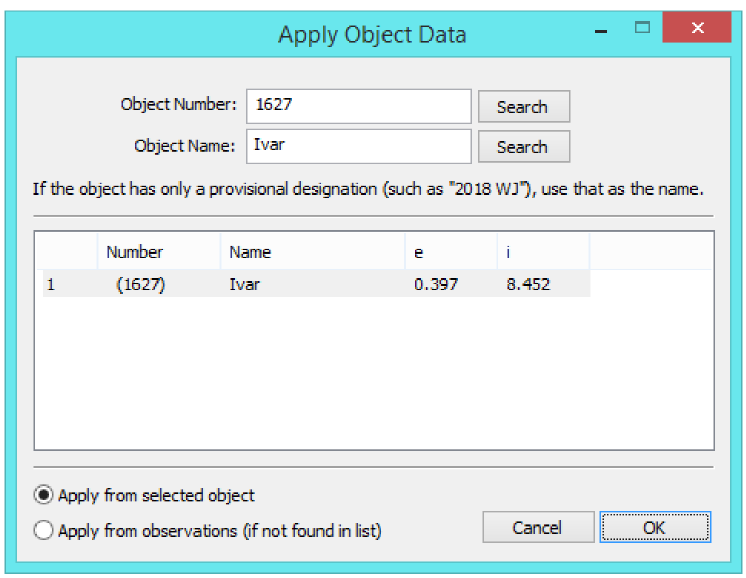


Figure 88 - Apply Object Data

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 for the *Find_Orb* instance associated with the object. If you do not already have such an instance 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 on "View in FindOrb". Navigate back to the "Select FindOrb Instance" window, and click the "Refresh" button to see the new instance appear in the list. Select it and then click "OK" to proceed.

After a moment, you should see the object data applied to each of the three photometry sets. 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.

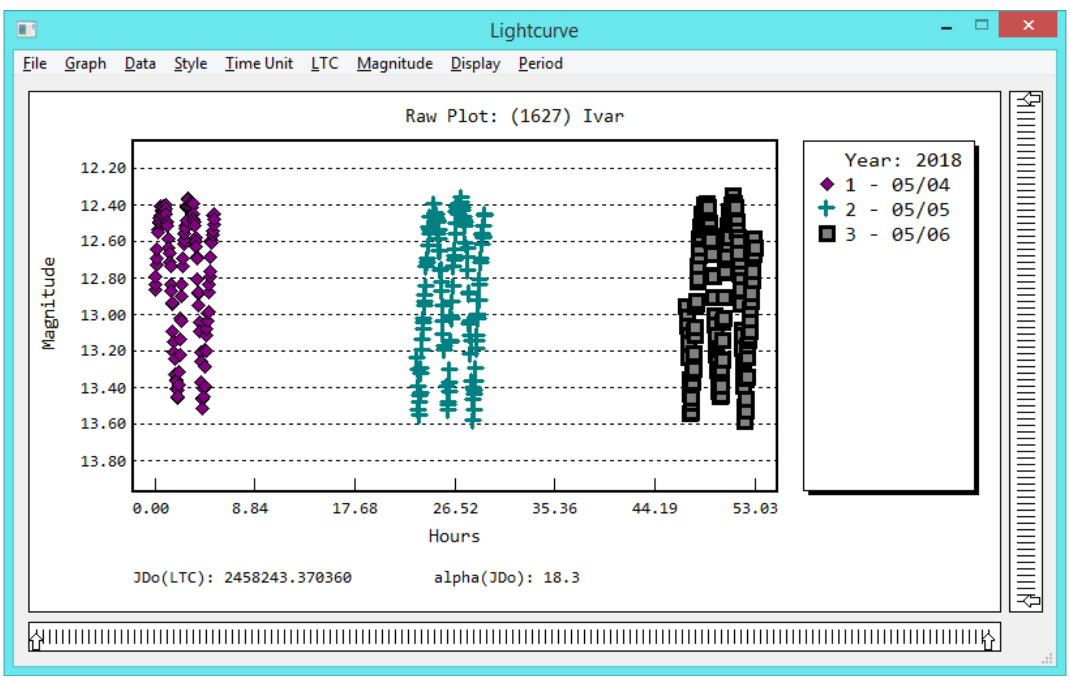


Figure 89 - Raw Plot of 3 Nights

Step 9: Final Period Search

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

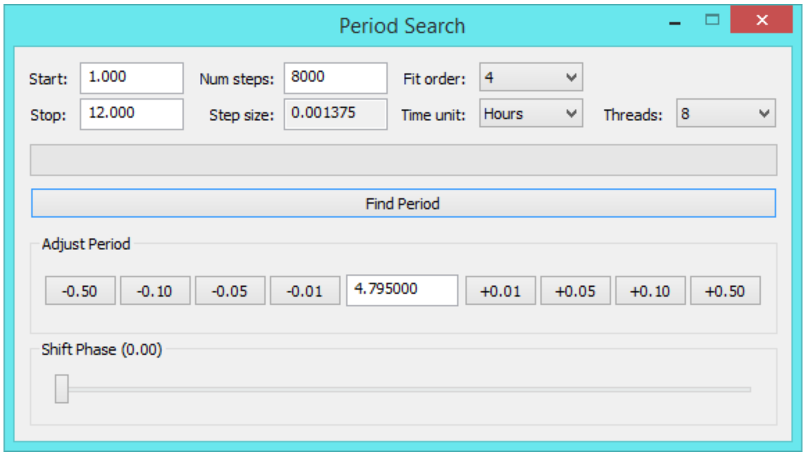


Figure 90 - Period Search Settings

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.

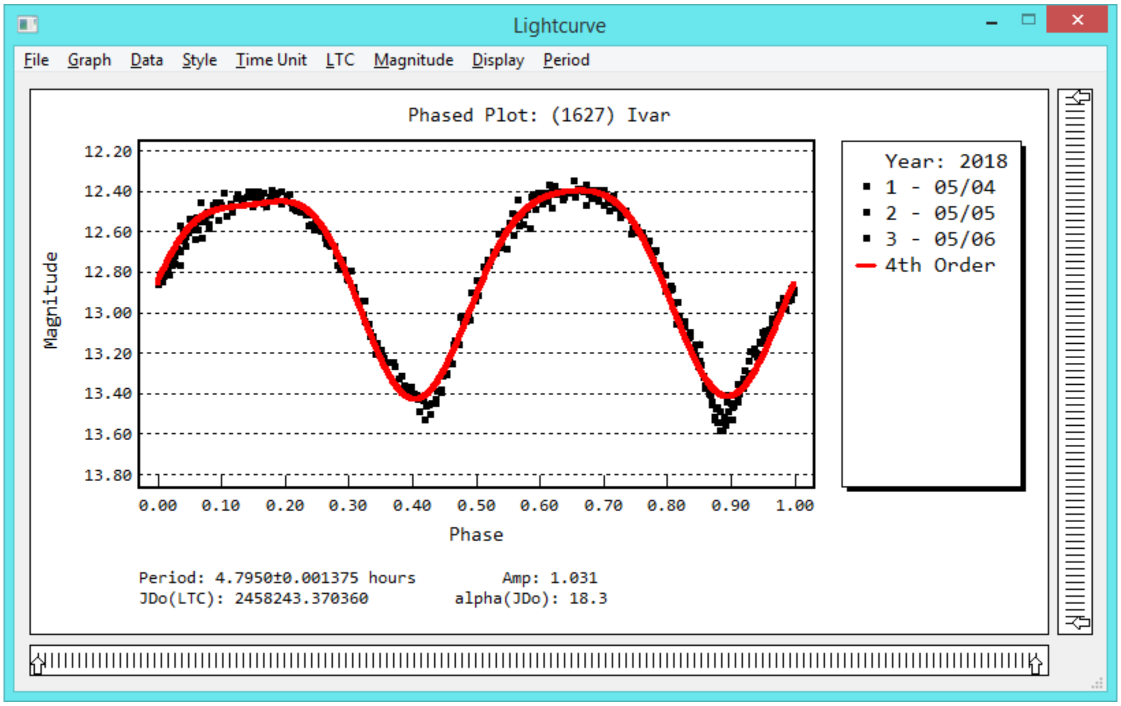


Figure 91 - Phased Plot of (1627) Using 3 Nights of Data

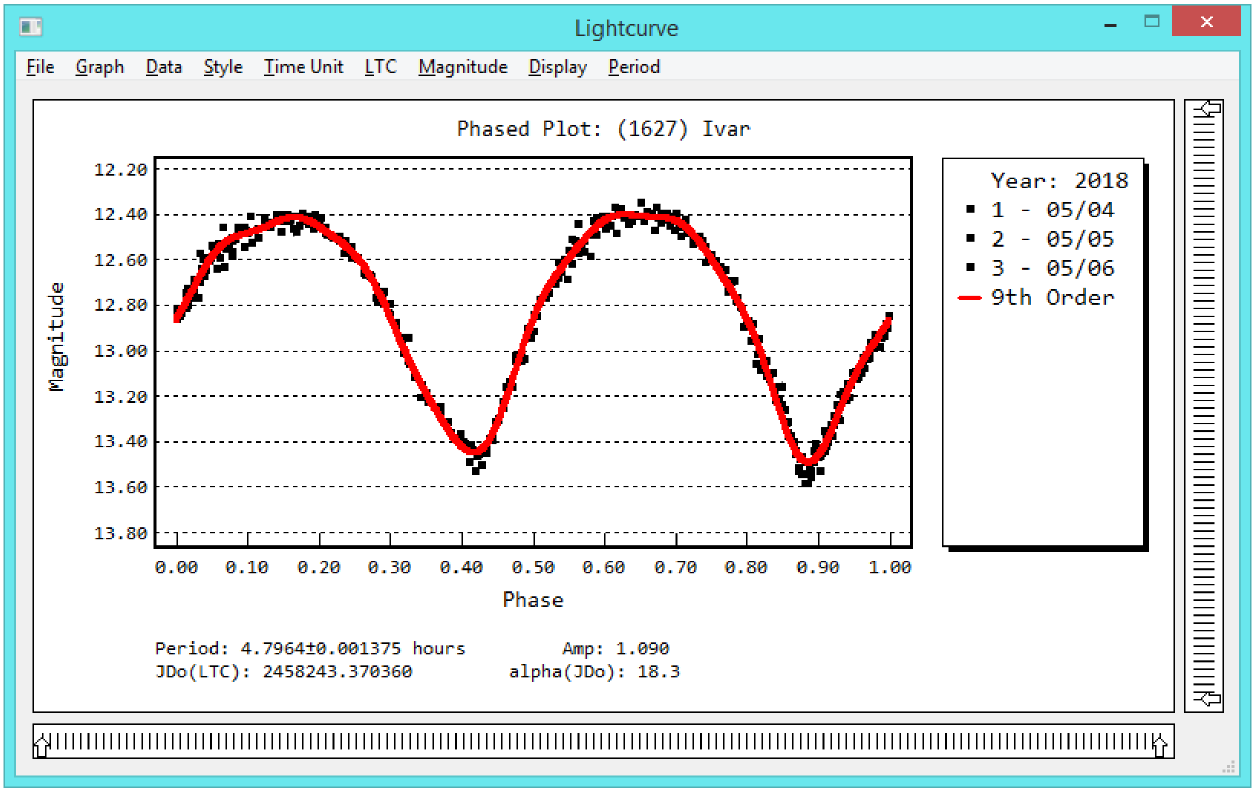


Figure 92 - Phased Plot using a 9th Order Fit

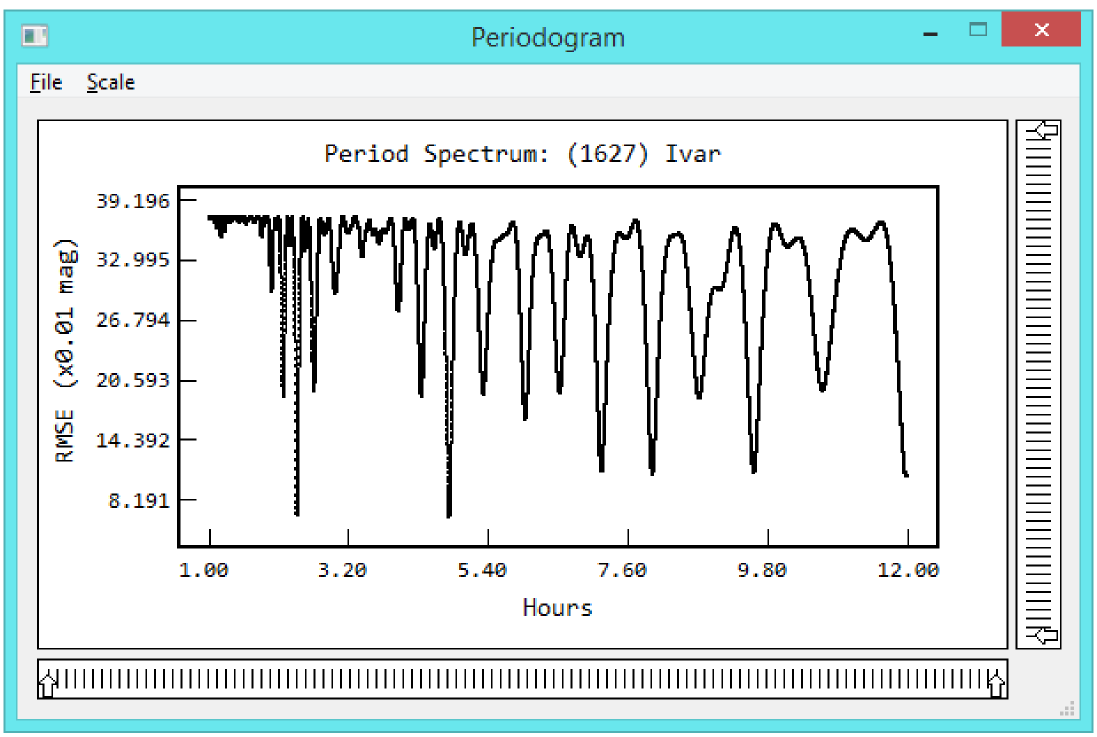


Figure 93 - Periodogram of (1627) with Search Interval of 1-12 Hours

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

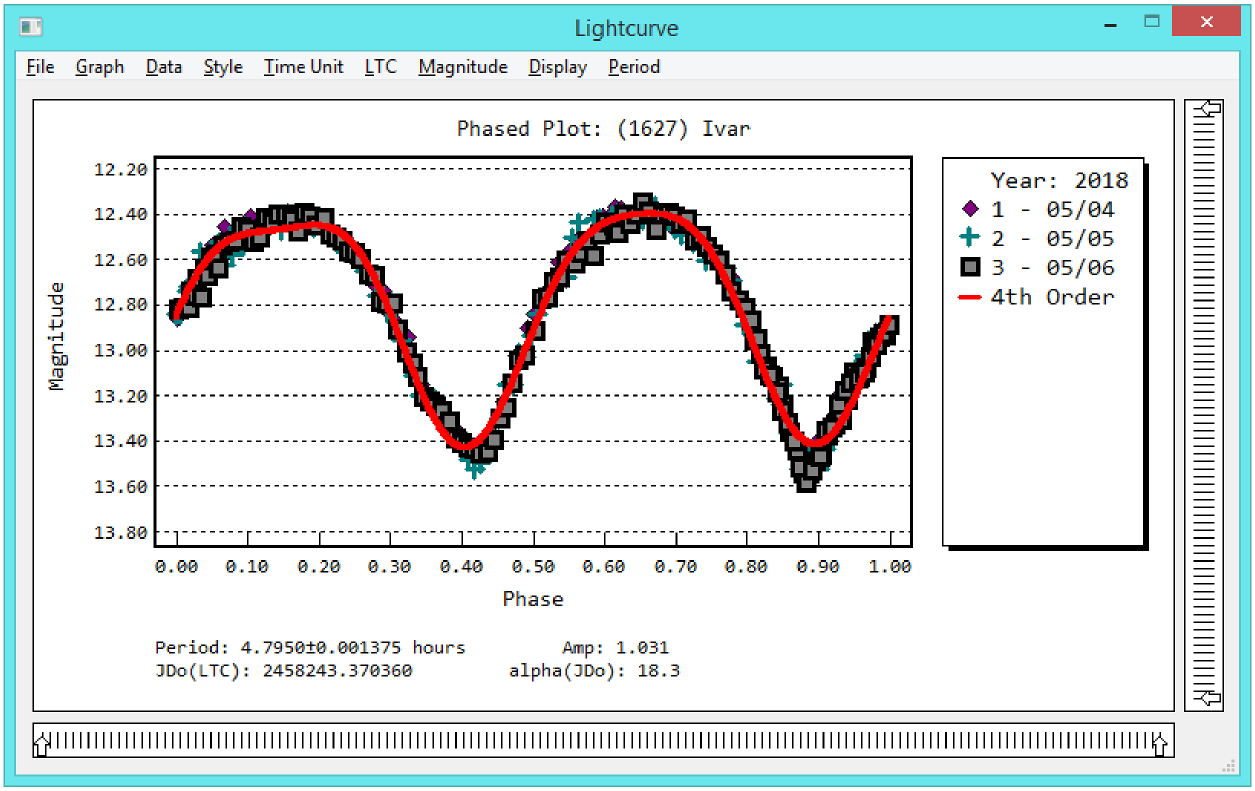


Figure 94 - Phased Plot using the Classic Style

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.

Lightcurve Database

Diam (km)	H	G	Albedo	Period	Amp	NumLC
7.500	12.990	0.240	0.2000	4.795000	1.480	50

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.

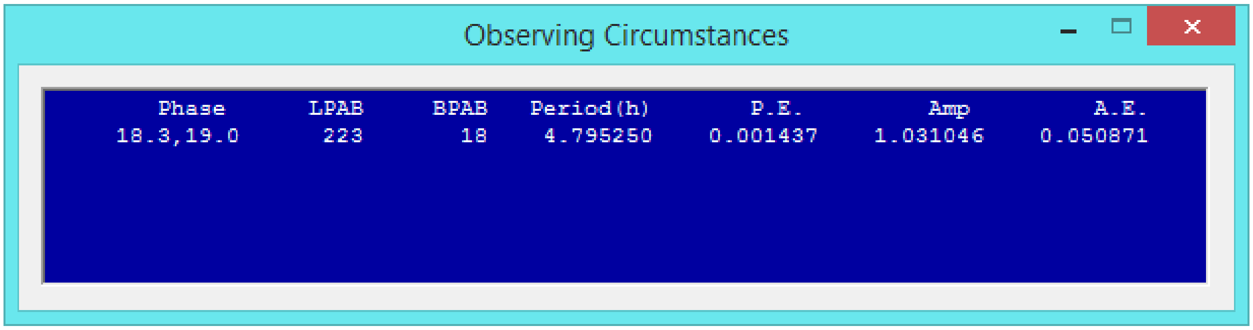


Figure 96 - Observing Circumstances Table Values

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 the orbit information returned by *Find_Orb* will be associated with that of the observatory referred to by the "MPCCode" column. If the code is 247 (indicating roving observer), then it 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 data, you will note that it has no object data attached to it. To apply object data, select the sets that are in the list and then right-click and choose "Apply object data". As before, you will be prompted for the *Find_Orb* instance that has computed the orbit for your object. Once object data has been applied, you will then be able to use the 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.

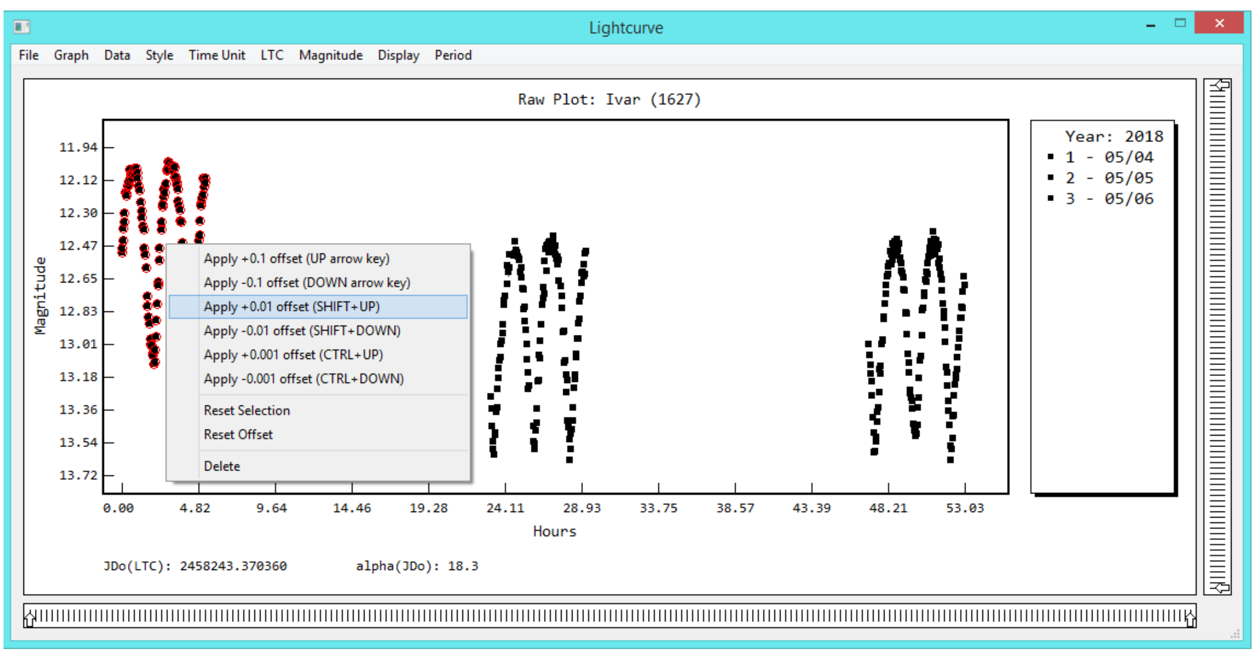


Figure 97 - Adjusting Magnitude Offsets

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. Next, choose *Action->Align Images* from the main menu.
4. Load the aligned images into the "Image Manager". Tycho saves the output images into a new directory with suffix of "_a", adjacent to the original directory.
5. Plate solve the images. Choose *Action->Plate Solve Images* from the main menu.
6. View the images. Choose *Action->View Images* from the main menu.
7. 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.
8. Open the "Generate Transforms" window. Choose *Photometry->Generate Transforms* from the menu of the "Image Viewer". Refer to Figure 99.
9. Select the desired filter for "Use Images having filter label", and the corresponding "Designated Filter". Normally, these two selections will be identical, meaning that "B" filter label corresponds to B filter. 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.
10. 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.

11. 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.

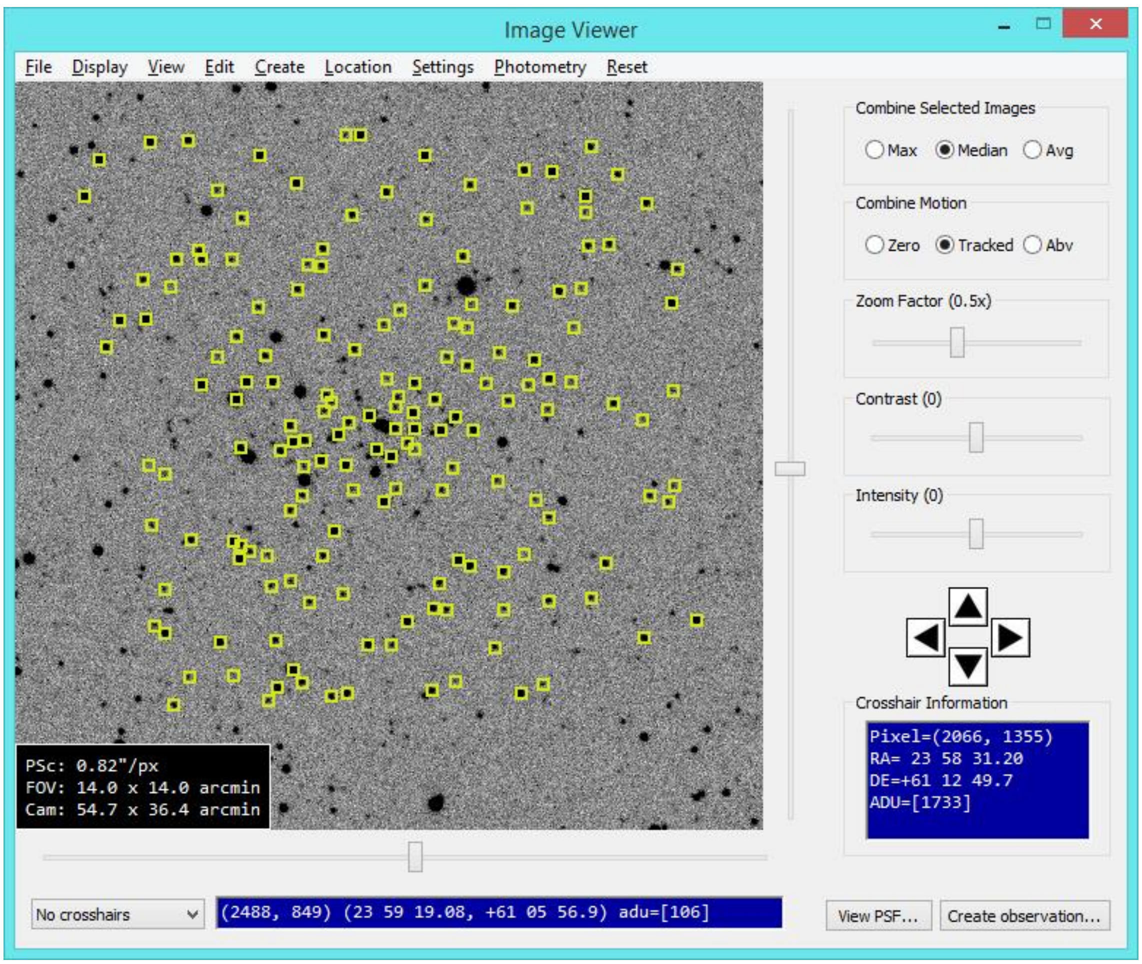


Figure 98 - Standard Field NGC 7790

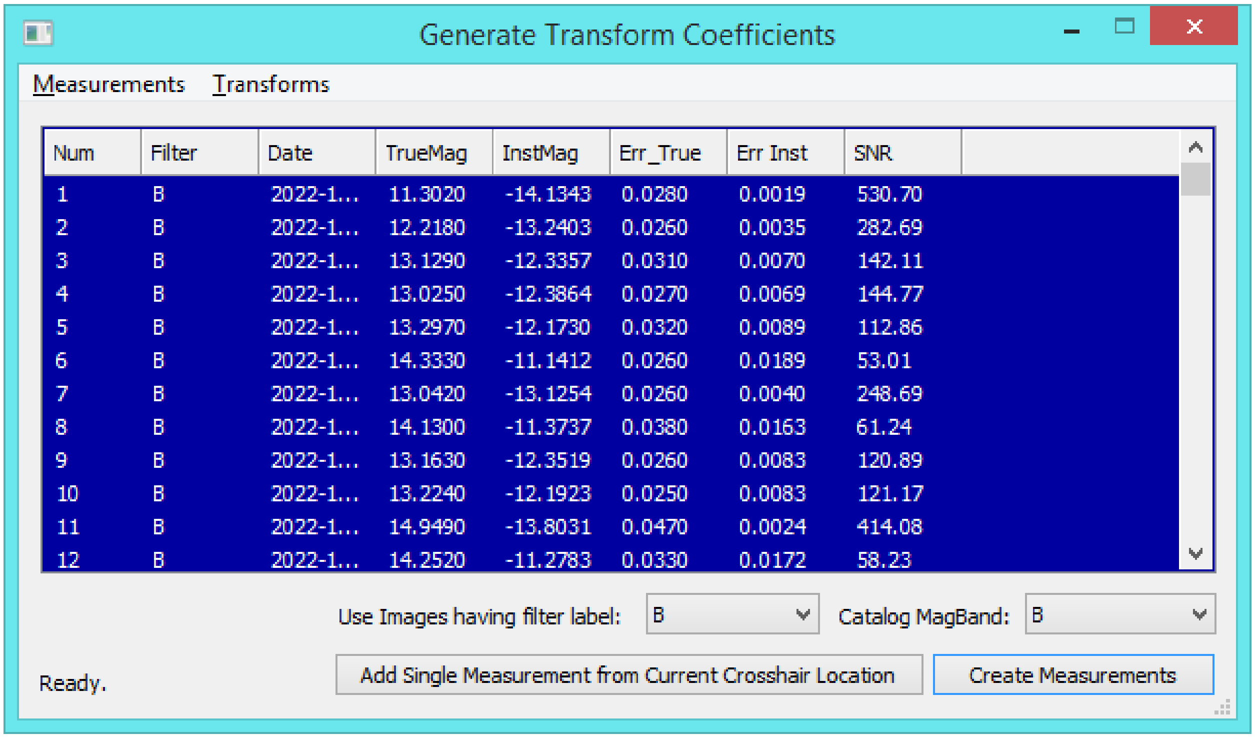


Figure 99 - Create Measurements

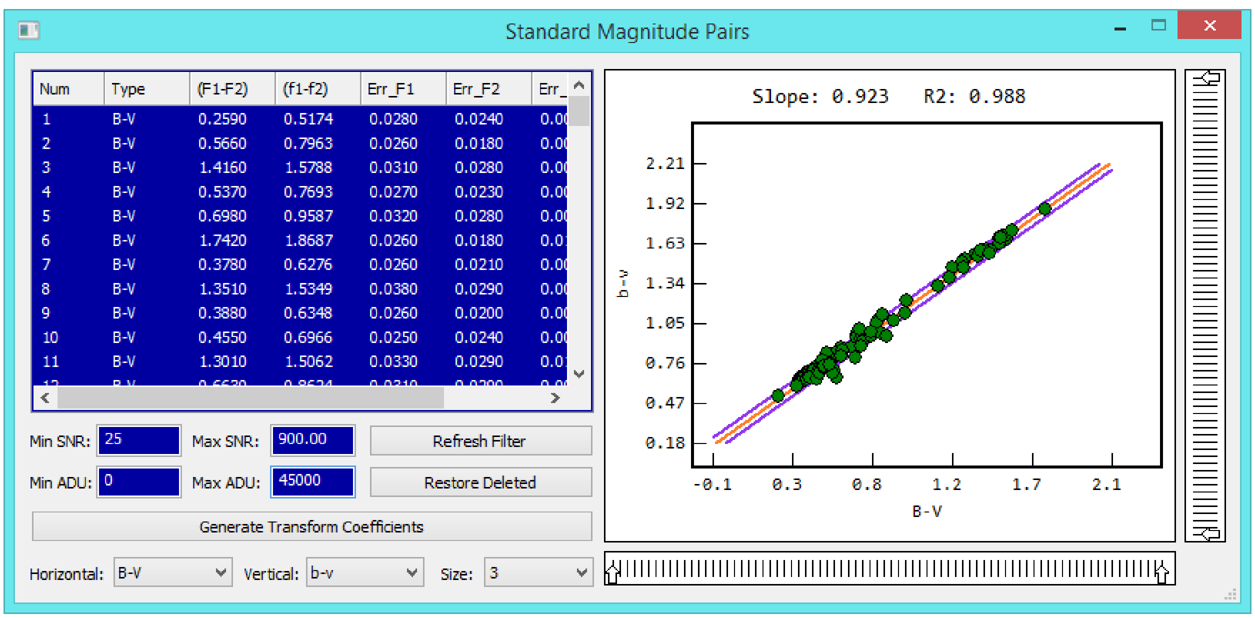


Figure 100 - Standard Magnitude Pairs

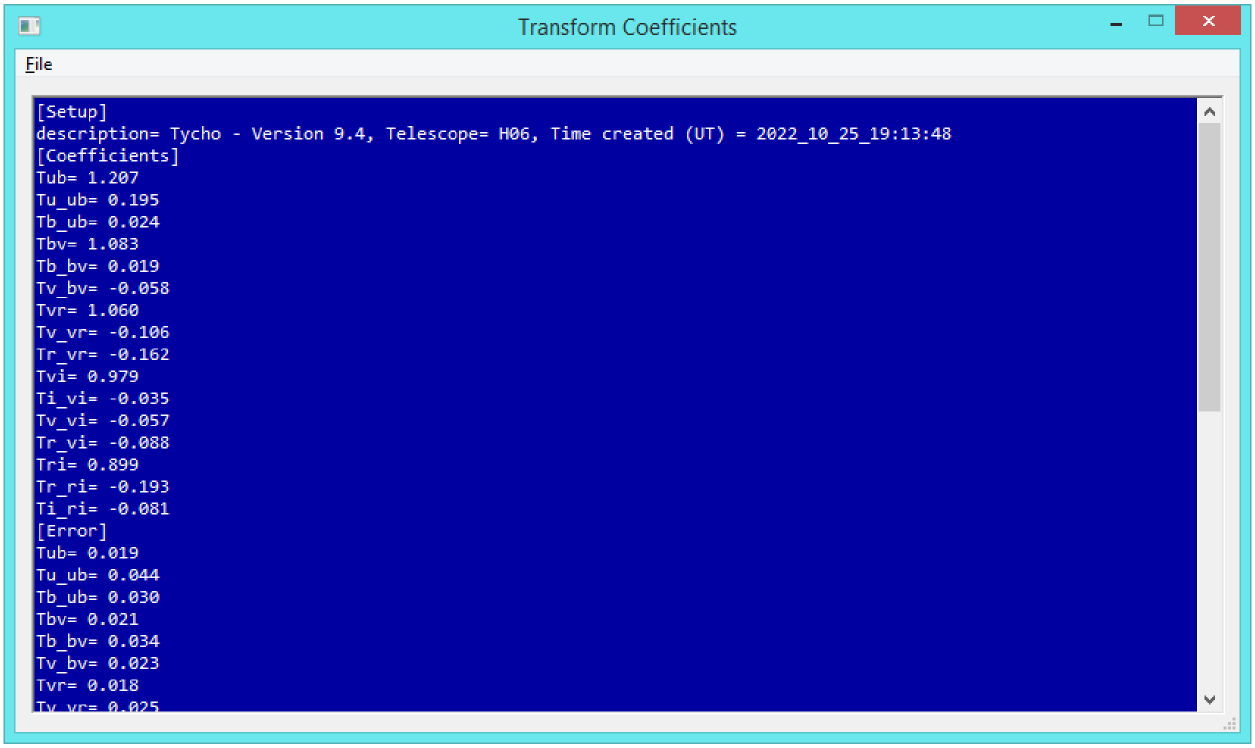


Figure 101 - Computed Transformation Coefficients

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. An example dataset can be downloaded here:

https://storage.googleapis.com/tycho_data-1/2/NGC7790_iTelescope_T11.zip

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 *Ephemeris->Attach from JPL Horizons* from the menu of the Session Planner. This will bring up the JPL Horizons interface, refer to Figure 102. 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 103.

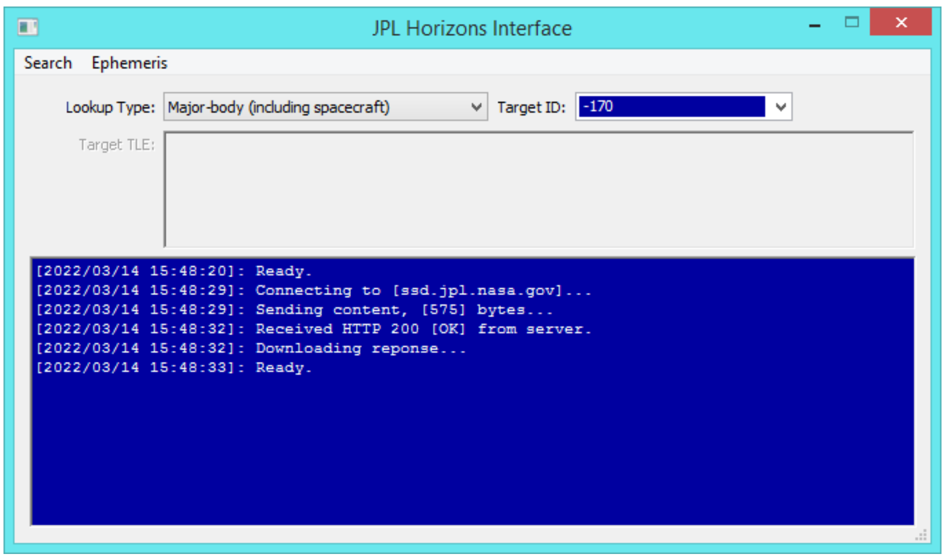


Figure 102 - JPL Horizons Interface

This will bring up a new window initially populated with every single major-body listed (482 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.

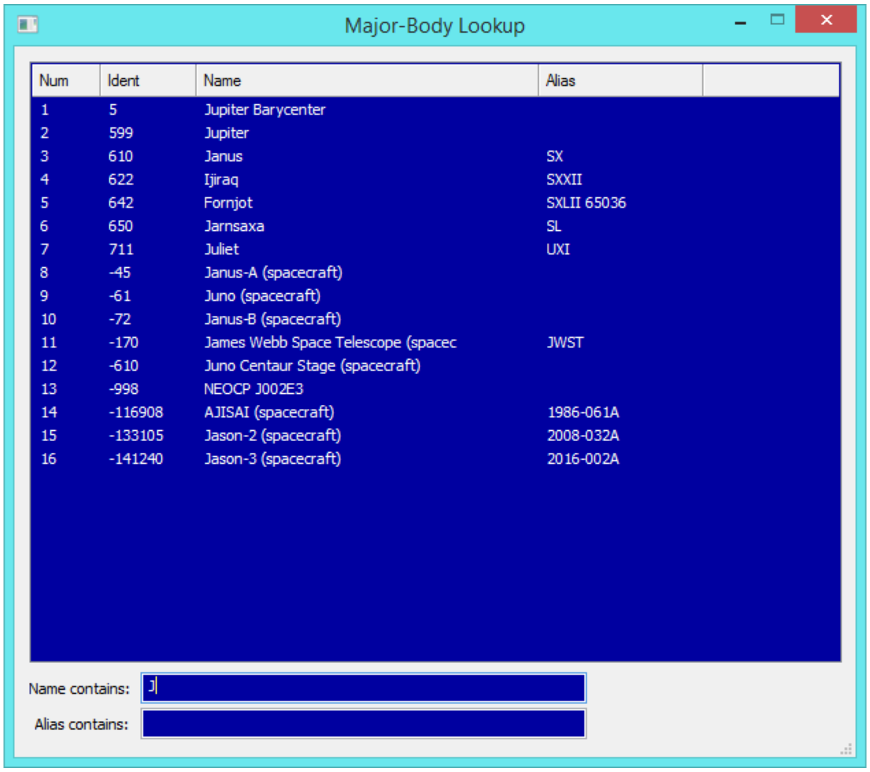


Figure 103 - Major-body Lookup for JPL Horizons Interface

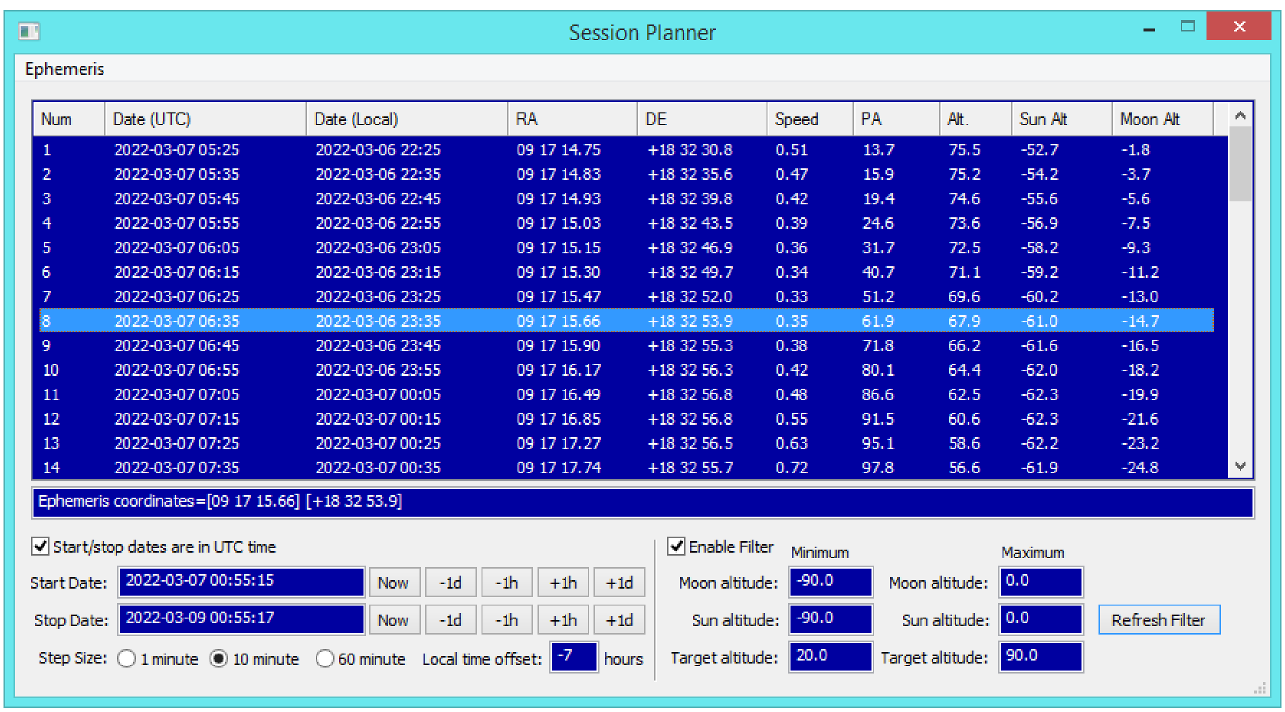


Figure 104 - Session Planner

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 104 for an example.

Here you can see that the "Start Date" is March 7, 2022 and the "Stop Date" is March 9, 2022. 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 104, 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 Find_Orb, 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 session plan by choosing *Ephemeris->Attach from Find_Orb* from the Session Planner menu.

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.

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 5 arguments
Usage: <tycho> <mode> <cpu> <input dir> <output dir>
<tycho> = Full path to Tycho executable
<mode>  = 12
<cpu>   0=use existing settings, 1=force CPU mode
<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 12 arguments
Usage: <tycho> <mode> <cpu> <width> <height> <hcrop> <vcrop> <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
<hcrop>  0=no cropping in horizontal; otherwise, specifies desired horizontal cropping
<vcrop>  0=no cropping in vertical; otherwise, specifies desired vertical 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
<align_mode>  0=fastest, 1=normal (do not use 'fastest' unless images are already mostly
aligned)
<dist_corr>   = Whether or not to apply distortion correction (0=no, 1=yes; only
available if align_mode=normal)
[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
```

Troubleshooting

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