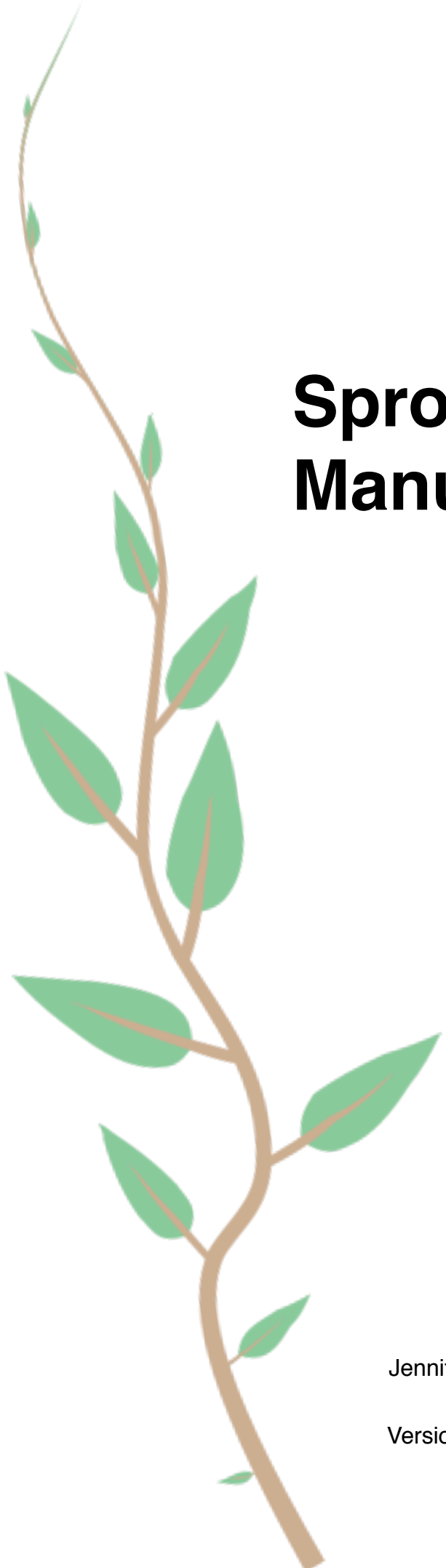


Sprout Manual

Jennifer Karr

Version 1.0; associated with code Version 1.0.

Introduction

Sprout is a program to generate model images of scattered light for circumstellar disks, in multiple polarization states, and to quickly and efficiently compare those models to observational data. The program has two parts: a GUI for setting up a grid of models, exploring parameter space in the disk geometry (varying the density, scale height and flaring of the disk), and a viewer, to interactively scroll through the models, comparing them with data.

Takami et al (2013) describes the physics of the code, and demonstrates its scientific applications.

This document contains instructions for using the Sprout GUI to run a grid of models, and then view the results. There are three main parts to the manual.

The first part provides basic installation instructions for the software.

The second describes how to use the GUI to generate a grid of models, including descriptions of the different parameters and details of the format for data output. The dust and disk models are briefly described (see Takami et al. (2013) for details of the model). The model is calculated following the method of Fischer et al, 1994.

The third describes how to use the viewer GUI to load and scroll through models, load and display observational data in contour, how to customize the plots and use different display modes, and how to save images and FITS files.

The model viewer is intended primarily for quick look and analysis; for publication quality plots or further analysis individual models can be read into a program to the users choice in either FITS or text format, as detailed in the manual, under *Output Formats* and *Saving Files*.

Illustrating figures are located at the end of the document. They are linked to the relevant section by hyperlinks (for those reading in digital form). For those reading from a hardcopy, keeping the figures and text sections as separate items lets you view both the description and the figure at the same time.

For comments, bug reports or questions, please contact Jennifer Karr and jkarr@asiaa.sinica.edu.tw.

Installing the Code

The code is written in a combination of C and python, for portability and affordability. It has successfully been installed on several flavours of Linux and OSX 10.6 (Snow Leopard). Feedback for installation on other platforms would be appreciated.

Software Requirements

You need python 2.6 or 2.7 and swig installed. In addition, the python packages Tkinter, pylab, matplotlib, numpy, scipy, pyfits, Pmw, pickle, os, string and sys are called. To test for installed packages, the included script "test_packages.py" can be used.

> ./test_packages.py

which will produce a list of any packages that need to be installed. To test for the presence of swig

> which swig

will determine if you have it installed. If not, it can be downloaded at www.swig.org,

Installing the Code

The code can be downloaded at http://www.asiaa.sinica.edu.tw/~jkarr/Sprout/sprout.html. On a *nix system (including OSX) the following commands will compile the C code and link it with the python code.

> gunzip sprout.tar.gz
> tar -xf sprout.tar
> cd Sprout_Code
> swig -python ccodes.i
> ./setup.py build_ext --inplace

This will place the files in the local directory. If you are redoing the installation for some reason, then enter

> rm -rf build _ccodes.so ccodes_wrap.c ccodes.py *pyc

to clear out intermediate files before re-executing the swig command. Once the program is compiled and running, you can delete the source code and created installation files.

If you want to put the files in another location

*> cp *.py dust_library.pickle Install_Directory/*

where Install_Directory is the directory you want to place the software. Make sure that the Install_Directory is defined in your path. You can then delete the source code and intermediate files:

*>rm -rf *.c build/ *.h _ccodes.so ccodes.i*

Executing the python setup script, test_packages.py and the Sprout programs will automatically use the python installation from your PATH variable. If you want to over-ride this with a specific version of python, explicitly run the programs with that version of python, instead of running the python as an executable. E.g.,

> python2.7 test_packages.py

Demo Files

[2]

Included with the code is a set of example files. The directory “Model_Example” contains a small run of data, to test the use of the view_models.py. The file “Model_Example.pkl” contains a save file for the parameters, that can be loaded into run_models.py. psf_example.fits contains an example PSF (derived from Subaru HICIAO observations). The file “test_data.fits” contains a FITS file that can be loaded into the viewer.

[3]

In addition, there is the script make_movies.sh (described in the “Making Movies” section of the manual).

[4]

Running the Models

Starting the Model

Start the model by running the python script from the command line (make sure your paths are set correctly.

> run_models.py &

General Structure

Figure 1: Basics of the Model GUI

The top line of the window has two buttons.

Quit: Quit the program without saving or running.

Help**:** Bring up a help file.

The second line of the window has functions related to running the model.

Run Simulations: This checks the input parameters for validity, displays a confirmation pop-up showing the size of the images in pixels, and the number of models to be run, and then asks for the output directory, before running the models.

Save Parameters: Save the current parameters to a file for future use.

Load Parameters: Load saved parameters.

Check Input: Check for non-valid entries (check for numerical input, maxima greater than minima, and positive values or integers where appropriate). Any errors will be displayed.

Save Results as Text: If checked, in addition to saving in python native pkl format, a portable ascii text file will be saved for each model (details of the file format can be found in the Output Format section below.)

On the third line, there are four tabs, for the different types of parameters.

Model Parameters: values dealing with the image size and resolution, number of photons, etc; the overall parameters that are independent of the geometry of the disk and physics of the dust grains.

Dust Parameters: Dust grain model and wavelength of the image.

Disk Parameters: Parameters related to the disk geometry; size, density, flaring, scale height, inner hole.

Advanced Parameters: Parameters of interest only to the advanced user.

Setting the Model Parameters

Figure 2: The Model Parameters Tab

The "*Model Parameters*" tab include values to do with the size, S/N and resolution of the output data. The units of resolution are strongly linked to the data to which you want to compare the models: the distance to the source, and the pixel scale of the observations.

Number of Photons (*1e6 photons*): a higher value gives a better S/N, but takes longer to run. 1e6 is a good starting value, you can go as low as 1e5 for quick look images. Very low density disks (ie, optically thin) will require more photons.

Viewing Angle (*degrees*): In addition to the ten viewing angles, one user specified viewing angle will be calculated. This is particularly useful if you know the inclination of the disk.

The viewing angles that are calculated by default are centred at 12.9, 31.4, 41.3, 49.4, 56.5, 63.2, 69.5, 75.5, 81.4 and 87.1 degrees, where 0 degrees is face on and 90 degrees is edge on.

Distance (*pc*): Distance to the source. The default value is Taurus (144 pc).

Pixel Sampling: Factor by which you want to scale the pixels in the models, compared to your original data. A value of 2 will use pixels twice as wide as the data. Rescaling the pixels can improve the S/N for a given run time, or speed up the model for a given S/N.

Pixel Scale (*arcsec*): Linear pixel size in the observational data. The default value is that for the PDI mode of HICIAO; change the value appropriate for your data. If you aren't comparing with data, this value can be set to whatever pixel size you want.

Name of Log File: Name of the file for the log information. This will be a text file, stored in the same directory as the model.

Note that the size of the image in pixels will be calculated from the pixel sampling, pixel scale and size of disk.

$$width\ in\ pixels = ceiling(2*radius/pixsize/distance/sampling*padding) \quad [1]$$

where *radius* is the radius of the disk, *pixsize* is the pixel size in the original data, *distance* is the distance to the source, *sampling* is the pixel sampling factor, and *padding* is a factor to add blank pixels around the image (useful for later convolutions). The size is rounded up to ensure that the value is odd, if necessary, for easy centring of the image. The padding is 10% by default, but can be changed in the advanced parameters (below).

Setting the Disk and Grid Parameters

Figure 3: Disk Parameters

The "*Disk Parameters*" tab covers the physical parameters for the disk model, including the range of values for the grid of models. The relevant equation for the disk is the Shakura-Sunyaev model (Shakura and Sunyaev, 1973). For a radius *r* in the midplane and a height *z* above the midplane, the gas plus dust density

$$\rho = \rho_o \left(\frac{r}{r_o}\right)^{-\alpha} \exp\left(-\frac{1}{2}\left[\frac{z}{h(r)}\right]^2\right) \qquad [2]$$

where the scale height

$$h = h_o \left(\frac{r}{r_o}\right)^{\beta} \qquad [3]$$

and

$$\alpha = \beta + 1 \qquad [4]$$

β is the flaring parameter, and ρ_o *and* h_o are calculated at the reference radius r_o.

Disk Minimum Radius (*AU*): The radius of the inner hole in the disk. A value of 0 gives no inner hole.

Disk Maximum Radius (*AU*): The radius of the disk.

Stellar Radius: ($R_\odot$) The radius of the central star. This parameter is only important for determining the values of ρ_o and h_o in units R*. If ρ_o and h are determined at an arbitrary radius in AU, R* is not used. (See "*Options for* h_o" and "*Options for* ρ_o" below for details)

Grid Values

For each of the grid values, you need to enter a minimum value, a maximum value, and the number of values. In addition, you can choose a logarithmic or linear scaling of the parameters; the endpoints will be included.

The three grid values

β: The flaring parameter in Equation [2].

ρ_o (*g cm^{-3}*): the initial density, measured at a reference point.

h_o (*R* or AU*): the disk scale height, measured at a reference point. h_o is measured in *R** if h_o is determined at *R**, and AU if h is determined at an arbitrary radius.

Options for h_o

The scale height h_o can be determined at R^* (the stellar surface), a common convention for this type of model, or the an arbitrary radius in AU. This can be selected under "*Options for h_o*". The default is R^*.

Options for ρ_o

There are two additional options for ρ_o. You can fix the total (dust) disk mask to a constant value (in solar masses). The program will scale the value of ρ_o in Equation 2 to satisfy this requirement. This is useful if you know the mass of the disk from other observations.

You can also scale ρ_o to a fixed value of the disk's optical depth in a given direction. In other words, at an angle of θ from the midplane, the total optical depth of the disk will be τ. This is useful for reducing the degeneracy in the parameters for a given shape of the disk emission.

Figure 4: Illustration of the optical depth vs angle option

Note that these two options for ρ_o will remove the grid option for ρ_o, as the density will be independently scaled in each model. The actual value of ρ_o used in the simulation will be stored with the model, used in the model file name, and displayed in the viewer.

Setting the Dust Parameters

Figure 5: Dust Parameters

These are options for the physical properties of the dust and the wavelength of the calculations. All the dust models are spherically symmetric homogenous particles, using silicate and either graphite or amorphous carbon.

Dust Model: a choice of a variety of pre-computed models taken from the literature.

KMH94, R_V=3.1
Cotera+01, amorphous
Cotera+01 x 15, amorphous
Cotera+01, graphite
Cotera+01 x 15, graphite

Cotera+01, amorphous (J98,400C)
Cotera+01, amorphous (J98,600C)
Cotera+01, amorphous (J98,800C)
Cotera+01, amorphous (J98,1000C)
Cotera+01 x 15, amorphous (J98,400C)
Cotera+01 x 15, amorphous (J98,600C)
Cotera+01 x 15, amorphous (J98,800C)
Cotera+01 x 15, amorphous (J98,1000C)

KMH94, R_V=3.1: ISM model taken from Kim et al., (1994)

Cotera+01: Grain composition based on circumstellar disk observations (Cotera et al., 2001), with variations in the calculation of optical properties.

- **amorphous** - amorphous carbon composition
- **graphite** - graphite composition
- **x 15** - the same composition, but with the size of the grains scaled up by a factor of 15, for more evolved dust.
- **(J98) 400C etc**. - The same composition, but the temperature indicates models with different pyrolisis temperatures, calculated in the manner of Jäger et al (1998).

Wavelength: Select the wavelength from the drop down menu: the choices are V (0.55 µm), J (1.25 µm), H (1.65 µm) and K (2.12 µm). This is a monochromatic wavelength, not integrated over a filter.

Advanced Parameters

Advanced parameters are for experienced users, and probably won't need to be changed.

For simulations of scattered light from disks, the dust is typically distribution in a very small angle above the midplane. Therefore, if we emit stellar photons isotropically, most of the photons will be lost without interacting with the disk. Either of the two parameters below can be used to limit the angle at which photons can be emitted, increasing computational speed. The scaling of the resulting images is adjusted to the expected total number of photons.

θ_{max} (*degrees*): angle above which photons are automatically considered to have escaped. Either θ_{max} or τ_{crit} can be set, but not both. If both are set, θ_{max} takes priority (-1 indicates the unset option). See Figure 7 for an illustration. θ_{max}=90 will emit photons isotropically.

τ_{crit}: optical depth through the disk below which photons are automatically considered to have escaped. Either θ_{max} or τ_{crit} can be set, but not both. If both are set, θ_{max} takes priority. See Figure 6 for an illustration.

Maximum Number of Scatterings: number of scatterings before a photon is considered absorbed. After a large number of scatterings, the photon loses energy and has little effect on the final image. Restricting the number of scatterings improves computational efficiency, and after N scattering events, the photon will no longer be tracked. The default number of three is good for quick look results; setting to 10 will give more accurate results for publication purposes. Setting it to 1 will show the contribution to the image from singly scattered photons.

Step size for Photon Path (*AU*): The grid used for integration in radius in the model. You may need to change this for very small disks (a smaller step size) or very large ones (a larger step size).

Range for Viewing Angle (*Degrees*): Angle over which the photons are averaged for the user specified angle.

Image Padding (*Percent*): amount of empty space around the edges of the model. 0 crops the model to the outer edge of the disk exactly.

κ_{ext}: ($cm^2 g^{-1}$) The opacity for the dust + gas. This will *override* the value used by the dust model, so use at your own risk.

Saving and Loading Model Parameters

Click "*Save Parameters*" in the second line of the window to save the model parameters to a file (you will be prompted for a file name). They can be loaded again using the "*Load "Parameters*" button. The parameter files are saved in native python pkl format.

Checking the Parameters

Click "*Check Input*" in the second line of the window. This will check the input for positive/ integer values, non-numeric values, and upper limits > lower limits. An error message will be displayed if there is a problem, a confirmation otherwise. This step will automatically be run before saving the parameters or running the model.

Running the Models

1) Click "Run Simulations"
2) The parameters will be checked for validity, and either confirm or give an error message. Close this window to proceed.
3) A popup window will list the pixel size of the output images, and the number of models to be calculated. If this is okay, click "Yes".
4) You will be prompted to choose a directory to store the output. Creating a new directory is strongly recommended. You *cannot* save more than one model in the same directory.
5) A popup window will show the number of models to be run, and an estimate of the remaining run time (after the first model has been run)
6) When the models are finished running, a popup will appear.

The Output

The output from the program will be contained in the chosen directory. It consists of a summary .pkl file for the run, and one file for each combination of ρ, h and β. Each file contains the information for all eleven viewing angles, and the four polarization vectors (I, Q, U, V) plus polarized intensity (PI). If the "*Save as Tex*t" option was chosen, the same data are also contained in text files (one per model). The Output Formats section below has a detailed description of the data format.

Units: The model images are scaled by the total incident flux from the star at the wavelength of the simulations. For comparison purposes, you can scale your observational data using observations of the star without the coronagraph or using the magnitude at the observational waveband to calculate the value. The above scaling does not include extinction, so if your star is heavily extincted it would be better to use the value for an unobscured star of the same spectral type.

Output Formats

Summary

The data are saved in a pickle file, a native python format, in the selected directory. There is a summary file for the run, and one file for each model, which contains all of the polarization states and viewing angles.

The file “run_info.pkl” contains the basic model information, and is primarily used by the viewer program.

Each model has a single pickle file which contains the values for each polarization state and viewing angle, with the name

"run_"+str("%5.3e" % h)+"_"+str("%5.3e" % rho)+"_"+str("%5.3e" % beta)+".pkl"

where *h*, *rho* and *beta* are the values for each model. (%5.3e indicates exponential notation with three decimal points). For example,

run_1.000e+01_1.000e-07_1.400e+00.pkl

is the file for a model with h_o=10, ρ_o*=1e-7, and* β*=1.4,*

In addition, selecting “*Save Results as Text*” saves each model in a text file, for ease of importation into other programs. A log file with a user specified name is saved in the same directory, with a summary of the run.

Individual Models, pickle Format

If you load the pickle file into python using the commands

```
>>> f = open(filename)
>>>dataObject=pickle.load(f)
```

then

dataObject = [data,twod,r_array,z_array]

where

data[keyword][angle_index]

is a two dimensional array of the image values values for

keyword="I", "Q", "U", "V", “PI”

indicating the polarization state, and

angle_index=0 to 10

giving the index of the viewing angle, with the user defined angle at the end of the array.

twod is a two dimensional array containing the projected view of the original density distribution, and r_array and z_array give the locus where the optical depth through the disk is 1 (used for density plots).

Individual Models, Text Format

The option "*Save as Text*" will also create an ascii text file for each model, for ease of importation into other software. The file name is the same as above, but with the extension ".dat".

run_"+str("%5.3e" % h)+"_"+str("%5.3e" % rho)+"_"+str("%5.3e" % beta)+".dat"

where *h, rho* and *beta* are the values for each model.

A series of header lines, prefixed by "##" containing a listing of the model parameters, followed by the data in the format

Angle	X	Y	I	Q	U	V
%12.4f	%5d	%5d	%12.4e	%12.4e	%12.4e	%12.4e

where *X* is the value of the x pixel (0 to *npixel*-1) and *Y* is the value of the y pixel (0 to *npixel*-1), both in integer format, followed by the four polarization values in exponential format.

The lines are organized by angle, then by X pixel coordinate, then by Y pixel coordinate. There are therefore *11 * nx * ny* total lines of data.

Scrolling through Models

If you've run a number of sets of models, there is a simple graphical tool to check the contents of a model directory: search_directories.py.

Figure 8: search_directories.py

1) Start the program

   > search_directories.py

2) Click "*View Directories*"
3) Choose a directory associated with a model
4) The window will show a quick summary of the parameters for the model, and the range of the variables.
5) Click "*Save to File*" to save the displayed information to a text file.

Using the Model Viewer

Loading and Viewing a Model

1) Start the Program

    > view_models.py &

2) Click "*Load Simulation*". Choose the directory which contains the model you want to view.
3) Choose the Image Mode you want to view from the drop down menu (see Figure 9).
4) Use the scroll bars to interactively change the viewing angle, scale height, initial density and flaring power. The plot will update as you drag the sliders.
5) At any point, clicking "*Plot/Refresh Plot*" at the bottom of the window will update the plot.
6) The "Plot Window Controls" (Figure 11) can be used to zoom in on sections of an image.

To Compare with Observational Data

The simulations are calculated to be easily compared to real data. The observational data must be in FITS format (with a single extension), should be centred in the image and be at the pixel scale specified in *"Model Parameters*" in the "Pixel Scale" entry. It does not need to be rotated to a specific orientation; this can be done in the viewer.

To Load the Data

1) Click "*Load Observations*" and select a FITS file containing your data.
2) Select the "*Show Data Contour*" button.
3) The program assumes that the data are centred in the image: the image will be automatically cropped and scaled to match the size of the model.
4) The program assumes that the pixel scale is consistent with the values used to generate the models.
5) The contour levels used will be listed for reference in the "*Image Options*" tab, under "*Levels*".

To rotate the contours with respect to the image

You can rotate the data if it is not aligned with the axis of the model images. You have two options; interactive rotation and entering the angle manually.

1) To interactively rotate, left click anywhere on the image, and without releasing the button drag the pointer as if you were rotating around the centre. When you release the mouse, the plot will update.

2) To enter a rotation manually, enter the angle (measured counter clockwise) in the entry box (labelled "*Contour Rotation*"), and click the "*Plot/Refresh Plot*" button at the bottom of the window.

To Convolve with a PSF

The image can be convolved with a user provided PSF, or a Gaussian. The user provided PSF must be in FITS format, in the same pixel scale as the data and centred in the image, but does not need to be flux scaled.

To convolve with a Gaussian PSF

1) Select "*Convolve with PSF*"
2) Choose "*Gaussian PSF*" and enter the FWHM of the PSF, in units of pixels of the original data.
3) Click "*Plot/Refresh Plot*" at the bottom of the window.

To convolve with a user provided PSF

1) Select "*Convolve with PSF*"
2) Choose "*File PSF*" and enter the name of the FITS file containing the PSF (including extension). The PSF must be in the same pixel scale as the original data, and should be centred.
3) If the PSF is in a different directory from the code, use the full path to the file.
4) Click "*Plot/Refresh Plot*" at the bottom of the window.

Loading/Calculating the PSF only needs to be done once; further plots will use the same PSF.

To Display Polarization Vectors

1) Select "*Show Polarization Vectors*" to display an overlay of the polarization vectors for the model. To customize the view, see the following section.

To Customize the Image

The image, contours and polarization vectors can be customized in the "*Image Options*" tab.

Figure 12: Image Options - customizing your view

Note on Colour Names

When entering a colour, a name can be used ("white", "red" etc), or a hex code (#FFFFFF, #FF0000, etc). Any valid HTML colour name can be used.

To use a logarithmic scale for the image

1) Select "*Logarithmic Image Scale*"
2) Click "*Plot/Refresh Plot*" at the bottom of the window.

To change the upper and lower limits of the image scale

1) Slide the "*Lower Limit"* and "*Upper Limit*" bars to adjust the image scale.
2) The units are percentile - ie, 99% will ignore values outside the 99th percentile for the upper limit, 5% will ignore values below the 5th percentile.
3) The plot will automatically update as you slide the bars.

To change the colour map of the image

Figure 13: Colour Maps

1) Click "*Change Colour Map*"
2) Click "*Show Colours*" on the popup if you want to view images of the available colour maps.
3) Select your colour map.
4) Click "Done"
5) Click "*Plot/Refresh Plot*" at the bottom of the window.

To mask the centre of the image

1) Change the Entry box for "*Mask Size*" to the size of the mask in model pixels.
2) Click The "Plot/Refresh Plot" Button At The Bottom Of The Window.
3) A mask size of 0 will have no mask.

To Change the Polarization Vectors

To use fewer vectors

1) Change "*Vector Resampling*" to an integer larger than 1. A value of 2, for example, will plot every second vector (the vectors are re-sampled, not averaged, to avoid smoothing out structure).
2) Click "*Plot/Refresh Plot*" at the bottom of the window.
3) Note that if the image has been convolved with the PSF, the polarization vectors will be calculated from the convolved data.

To make the vectors longer or shorter

1) Change the "*Length Scale*" entry to a value less than 1 (shorter) or greater than 1 (longer). This will scale the length and width of the vectors.
2) Click "*Plot/Refresh Plot*" at the bottom of the window.

To Change the Colour of the Polarization Vectors

1) Enter an appropriate colour in the "Vector Colour" box.
2) Click "*Plot/Refresh Plot*" at the bottom of the window.

Customizing the Contours

To change the contour colours

1) Enter the colour name under the Contour Options section. This works with or without the "*Custom Contours*" option chosen.
2) Click "*Plot/Refresh Plot*" at the bottom of the window.

To generate custom contours

1) Click "*Data Statistics*" to view a histogram of the data, (this will help select physically reasonable values).
2) Enter the number of contours, the lower limit of the contours, and the upper limit of the contours in the "*Contour Options*" Section.
3) Click "*Generate*". A list of contours will be displayed in the "*Levels*" entry box.
4) Click "*Plot/Refresh Plot*" at the bottom of the window.

To enter your own arbitrary contours

1) Enter your contour values in the "*Levels*" Entry box, in ascending order, separated by commas (i.e. 0,1.0,2.5,5.0).
2) Click "*Plot/Refresh Plot*" at the bottom of the window.

To use logarithmic contours

1) Selecting "*Logarithmic Contours*" and refreshing the plot will automatically generate logarithmic contours.
2) Selecting "*Logarithmic Contours*" and using the "*Generate*" method, as above, will generate custom logarithmic contours. Negative values for the data minimum are acceptable.

Plotting Profiles

Figure 14: Viewing Radial Profiles.

If "*Radial Profile (I)*" or "*Radial Profile (PI)*" has been chosen for the viewing mode, the viewer will plot a radial profile calculated along the X or Y axis for either the total intensity, or the polarized intensity. The data can be averaged along a strip X pixels wide (model pixels), or along a wedge of constant angle X degrees wide. The contour data, if displayed, will be averaged in the same way.

1) Select "*Radial Profile (I*)" or "R*adial Profile (PI)*" from the drop down menu.
2) Select the "*Profile Options*" Tab.
3) Choose the axis you want to plot (see Figure 14 for definitions).
4) Choose *Wedge* (by angle) or *Slice* (by pixel) for the profile type, and set the width of the strip (in model units, for pixels).
5) Select the plotting symbols for the model and the data.
6) Enter colours for the model and the data.
7) Click "*Plot/Refresh Plot*" at the bottom of the window.

Customizing the Profile

1) You can choose a logarithmic plotting scale for the X or Y axis in the plot.
2) You can scale the data by an arbitrary value by entering a value in the "*Scale Data by*" entry box and refreshing the plot.

Plotting Density Distributions

Figure 15: Density Distributions

You can display a profile of the underlying density distribution, in r vs z coordinates.

1) Select the "Density Distribution" option from the drop down menu.
2) Click "*Plot/Refresh Plot*" at the bottom of the window.

Saving Images and Making Lots of Plots

You can save individual image to a file from the image menu (seen in Figure 11), with the format determined by the file name extension. In addition, you can plot multiple images in a single plot, and make plots for all the models in a run. The files will be saved in the same directory as the data.

Figure 16: Snapshot View - viewing multiple images

Snapshot View

The "*Snapshot View*" generates a grid of images in two parameters for a quick view of a region of parameter space. (e.g. plotting the angle from 12.9 to 49.4 on one axis, and β from 1.0 to 1.5 on the other)

1) Set up the image options the way you want it to appear: the type of image, image and contour options, colours, etc, as described in the above sections.
2) Select the "*Snapshot View*" tab.
3) Choose two variables by selecting one button in the first column, and one in the second.
4) Move the sliders for these two variables to choose the upper and lower limits.
5) Upper = Lower will produce a single image in that axis. Upper < Lower will display the images in reverse order.
6) Click "*Make Figs*". This may take a moment, particularly for many complex images.
7) Save using the menu in the image window.

Generating All Images

This option generates a single graphics file (jpg) for each model in the selected viewing mode. Images will have the same file name as the data files, with an extension for the angle and images mode, and will be saved in the data directory. If there is data loaded, it will also create an image file of the data named "data.jpg".

This can be a lot of images - one for each model, times 11 for the different angles, and can take a few minutes to generate.

1) Set up the image options the way you want it to appear: the type of image, image and contour options, colours, and so on.
2) Click "*Save All Figs"*
3) Wait a while.

Saving Files

1) Clicking "*Save Current File*" will save the current model and viewing angle as a FITS file, with five extensions; one each for I, Q, U, V and PI.
2) In the Profile View mode, selecting "*Profile Options*" and clicking "*Save Current Profile*" will save the currently extracted profile to a text file.
3) The files ill be saved to the same directory as the data.

Making Movies

The viewer doesn't have a built in function for making movies from multiple models, however, this isn't too hard to do using the ImageMagick package. The script "make_movie.sh" (provided) will generate a movie from a list of files using ImageMagick's convert function.

> *> ./make_movie.sh files.txt mymovie.mp4*

where *files.tx*t is a list of image files, one per line, and *mymovie.mp4* is the output file. (*mymovie.mov, mymovie.avi,* etc will also work).

Note that this is a fairly quick and dirty routine, and will write over existing files. The command

> *> convert -antialias -delay 1x3 *.jpg mymovie.mp4*

has a similar effect, taking the files in listing order. Changing 1x3 to 1x4 (or higher) will speed up the movie.

References

Cotera, Angela S.; Whitney, Barbara A.; Young, Erick; Wolff, Michael J.; Wood, Kenneth; Povich, Matthew; Schneider, Glenn; Rieke, Marcia; Thompson, Rodger, ApJ **556**, 958, 2001.

Fischer, O, Henning, Th, Yorke, H.W., 1994, A&A **284**, 178

Jäger, C., Mutschke, H., and Henning, T.: 1998, A&A **332**, 291

Kim, S-H, Martin, P.G., and Hendry, P.D., ApJ **431**, 783, 1994

Shakura, N. I. and Sunyaev, R. A.: 1973, A&A **24**, 337

Takami, M.,ApJ, submitted.

Figures

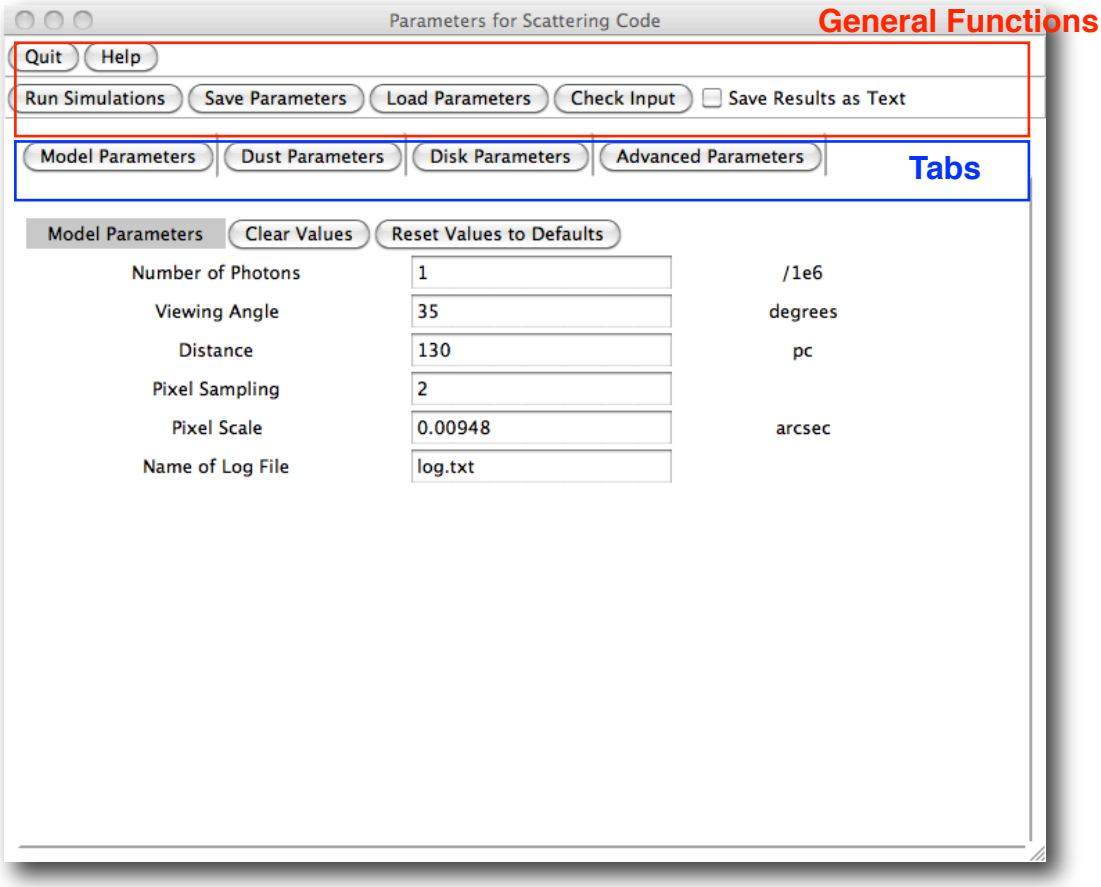


***Figure 1**: General Functions - illustration of the general functions of the GUI and the tabs for selecting the windows for different parameter sets. The general functions are always visible.*

[Back to Text]

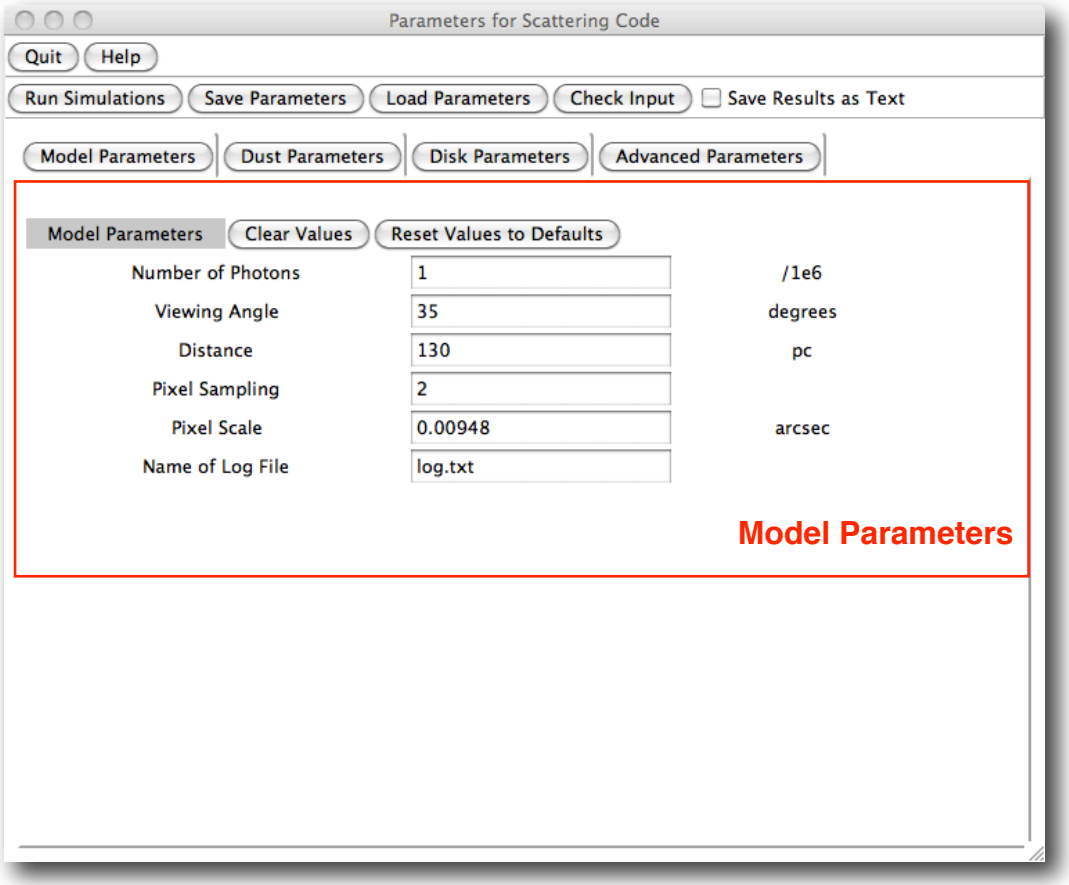


***Figure 2**: Model Parameters - values related to the running of the model. This example uses 10^6 photons and a user specified viewing angle of 35 degrees. The source is at a distance of 130 pc, and the model will use pixels twice the size of the data, or 2x0.00948"=0.01896". The log file will be written to in log.txt*

[Back to Text]

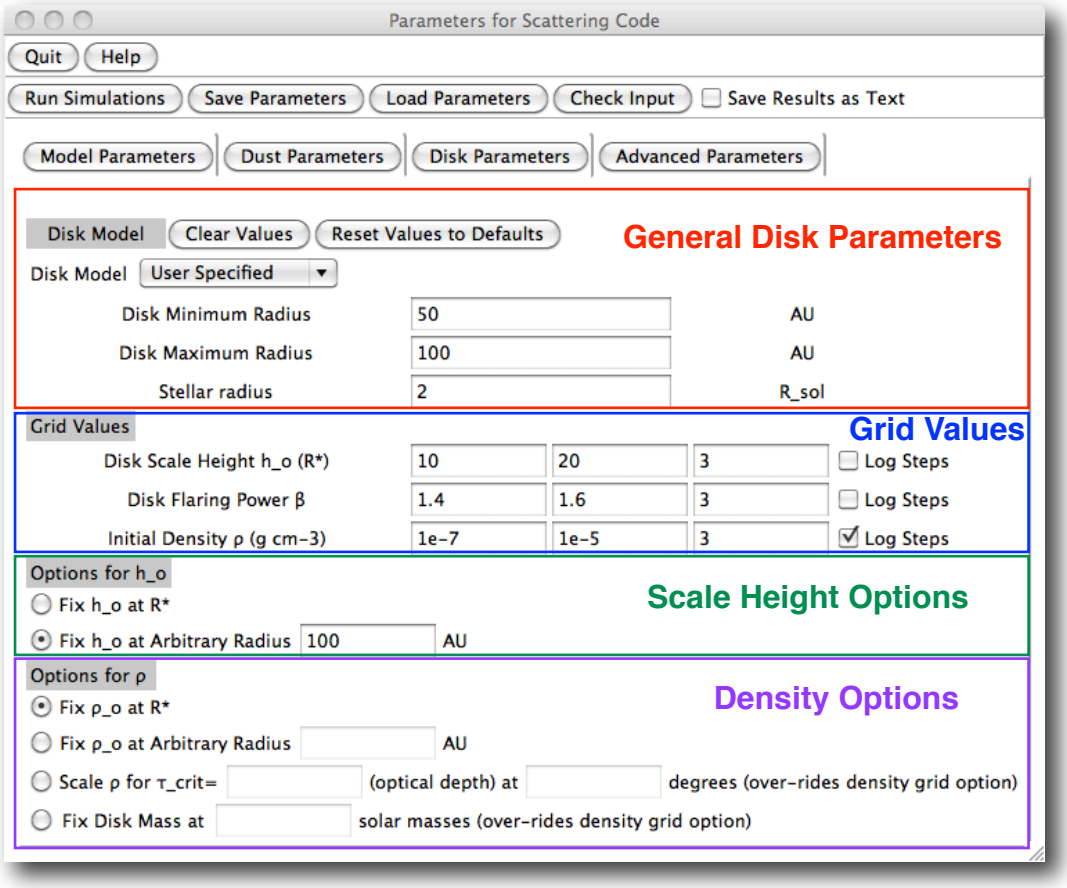


***Figure 3**: Disk Parameters - values related to the geometry of the disk. In this example, the disk is a ring like structure, with an inner radius of 50 AU, and an outer radius of 100 AU. The stellar radius is 2 $R_\odot$, which will be used to calculate the density, which is determined at R*. The scale height h is fixed at a radius of 100 AU. The grid of models will consist of 3 values in h; 10, 15 and 20 AU, 3 values in β; 1.4, 1.5 and 1.6, and 3 value of ρ, logarithmically spaced; 1e-7, 1e-6 and 1e-5 g cm^{-3}.*

[Back to Text]

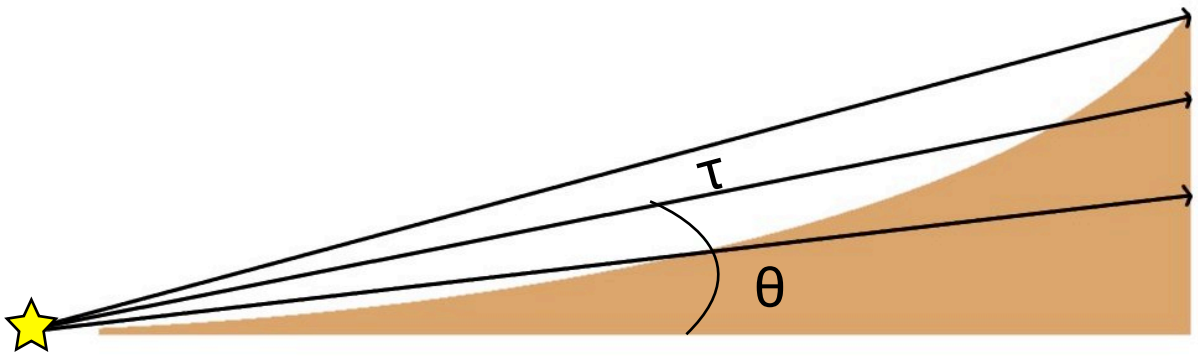


***Figure 4**: Illustration of the advanced density option. Each line traces a total optical depth through the disk, τ, at an angle θ with the midplane. The value of τ for a particular angle θ can be fixed, and the total mass of the disk and initial density will be scaled to this value.*

Back to Text

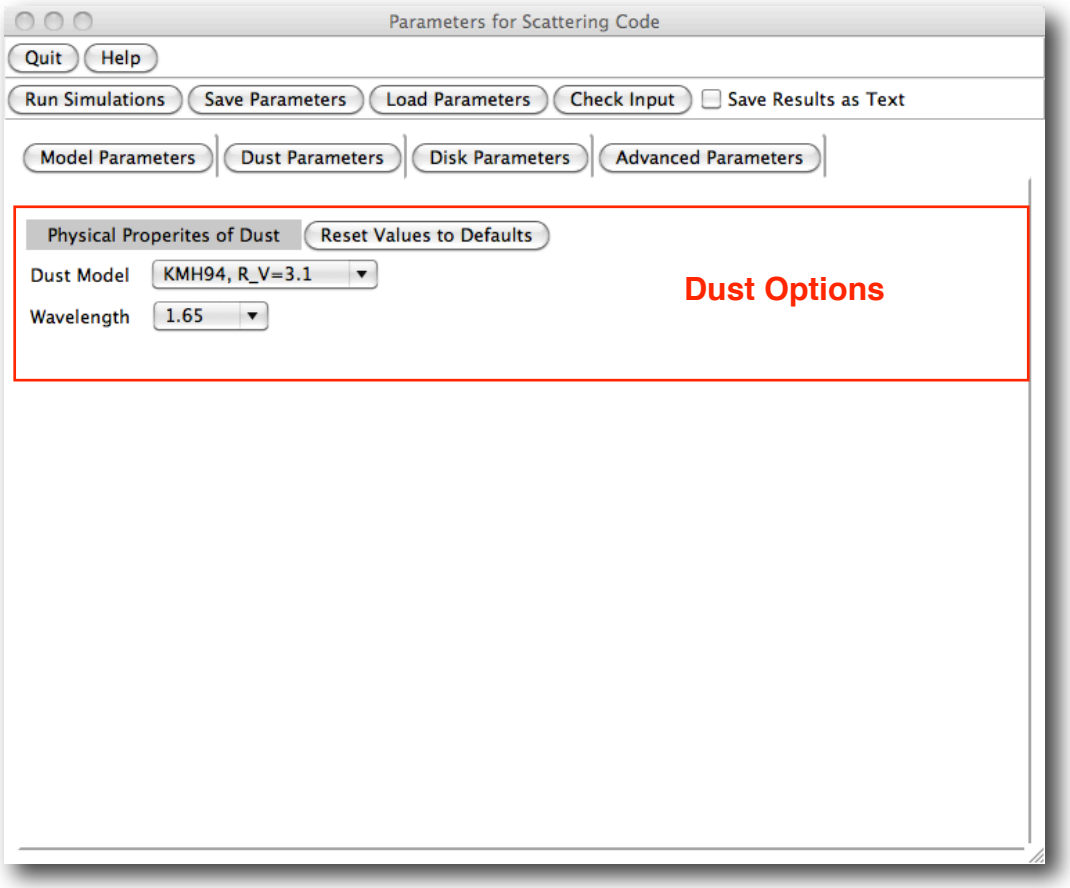


***Figure 5**: Dust Parameters - values related to the dust model. This model uses an ISM model with graphite and amorphous silicates, and is calculated at a wavelength of 1.65 microns (H band).*

Back to Text

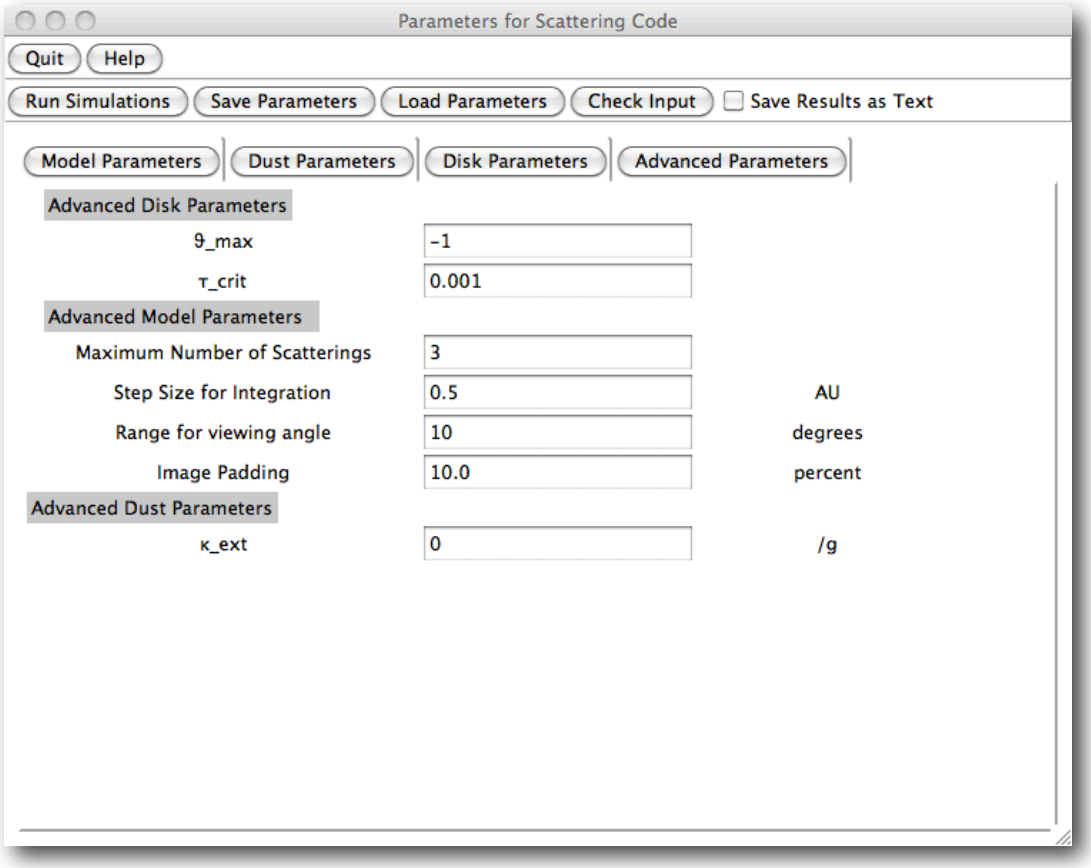


***Figure 6**: The Advanced Parameters tab, showing the default vaules.*

Back to Text

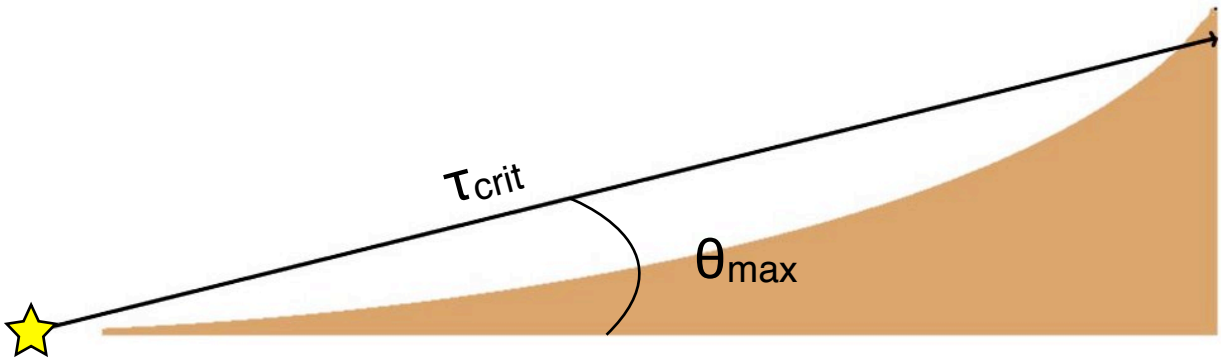


***Figure 7**: An illustration of the τ_{crit} and θ_{max} options. Photons emitted or scattered above this line, which forms an angle θ with the midplane, and has a total optical depth to the edge of the disk, will be considered lost. The parameter can be specified in either form τ_{crit} or θ_{max}.*

[Back to Text]

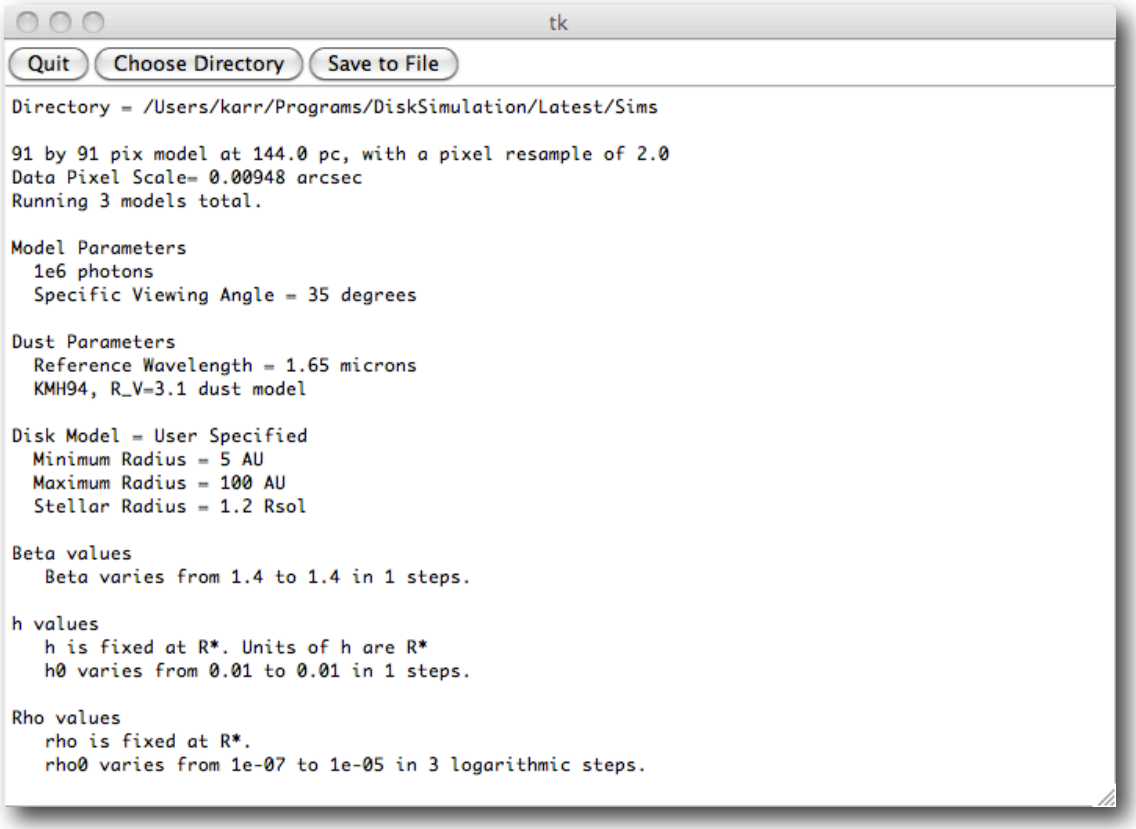


***Figure 8**: Searching the directories: an example showing the information for a set of models.*

Back to Text

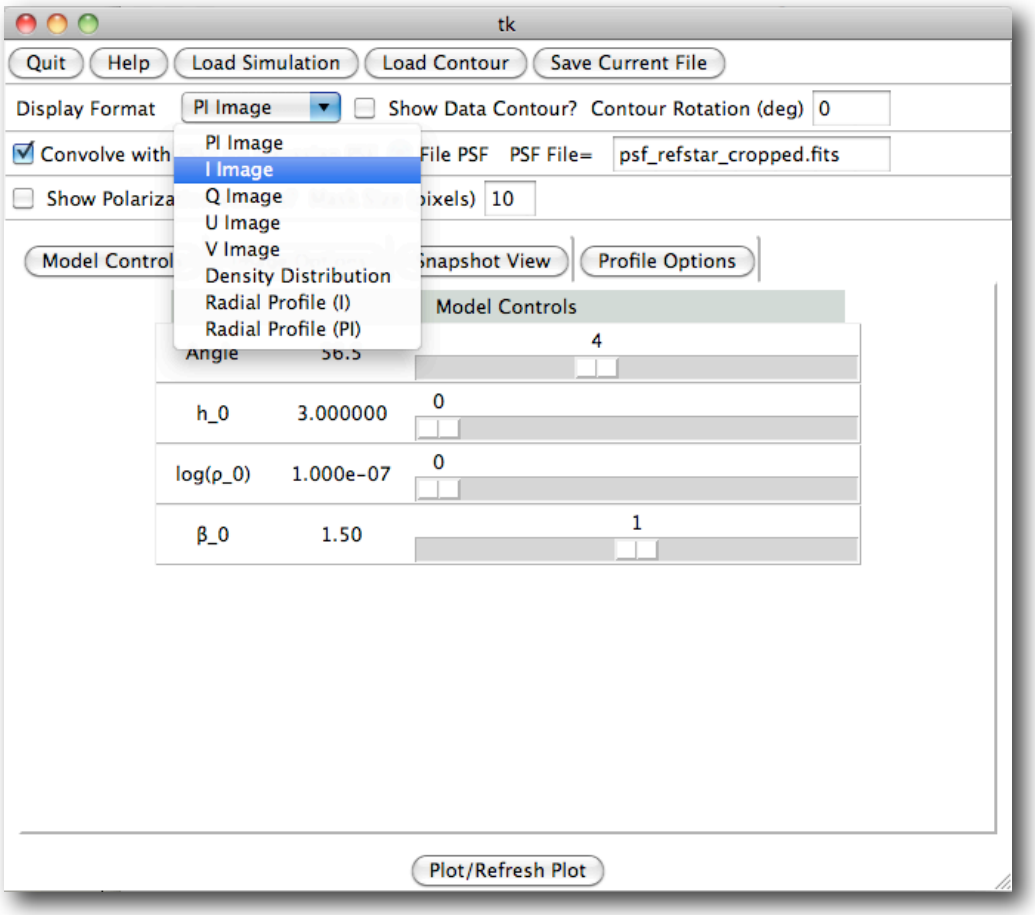


***Figure 9**: The model viewing interface - selecting a viewing mode; in this case, total intensity (I)*

Back to Text

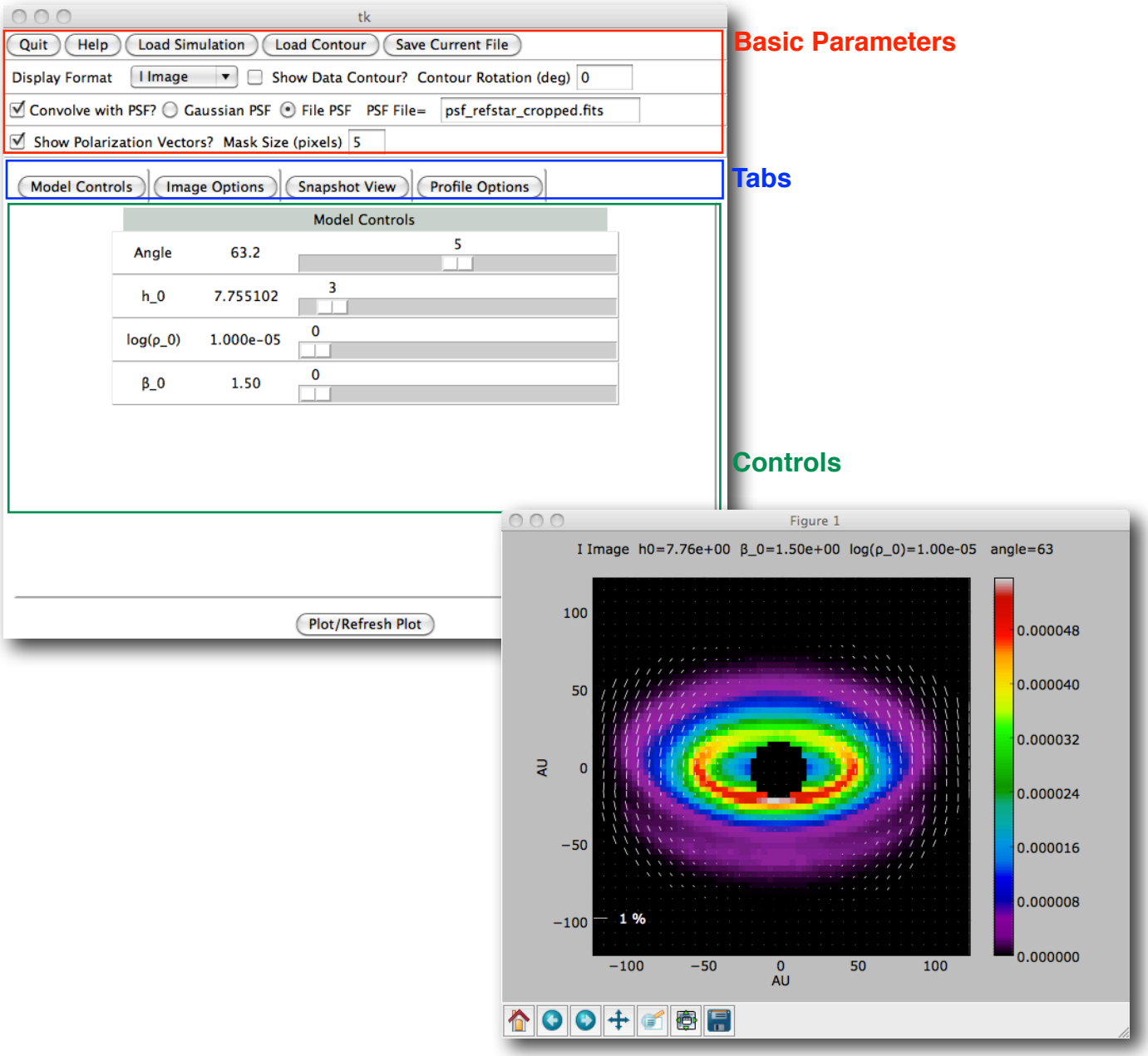


Figure 10: *Top: The model viewer window. The basic functions are at the top: loading the simulation and contour data, saving the current file, selecting the current display mode, the contour and polarization display options, contour rotation and mask size and options for convolving with a PSF. The tabs select different sets of options for customizing the view.*

Bottom: An example, showing an intensity image for a model with h_o=7.75 R, β=1.5, ρ=1e-5 cm-3, at a viewing angle of 63 degrees (matching the window above). Polarization vectors are shown, and the image has been convolved with a user provided PSF. The central 5 pixels of the image have been masked.*

[Back to Text]

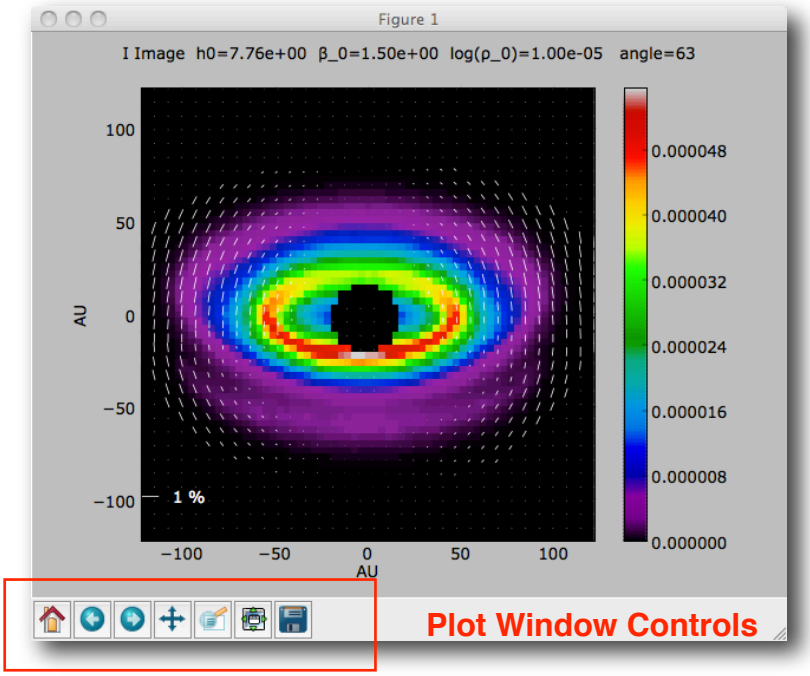


***Figure 11**: The Plot Window Controls - On the lower left of the plot window are built-in controls to adjust the plot. From left to right, return to the original image, move left, move right, pan, zoom in on a selected region, configure plot, and save image to file. The last of these can be used to save an individual view to a graphics file.*

[Back to Text]

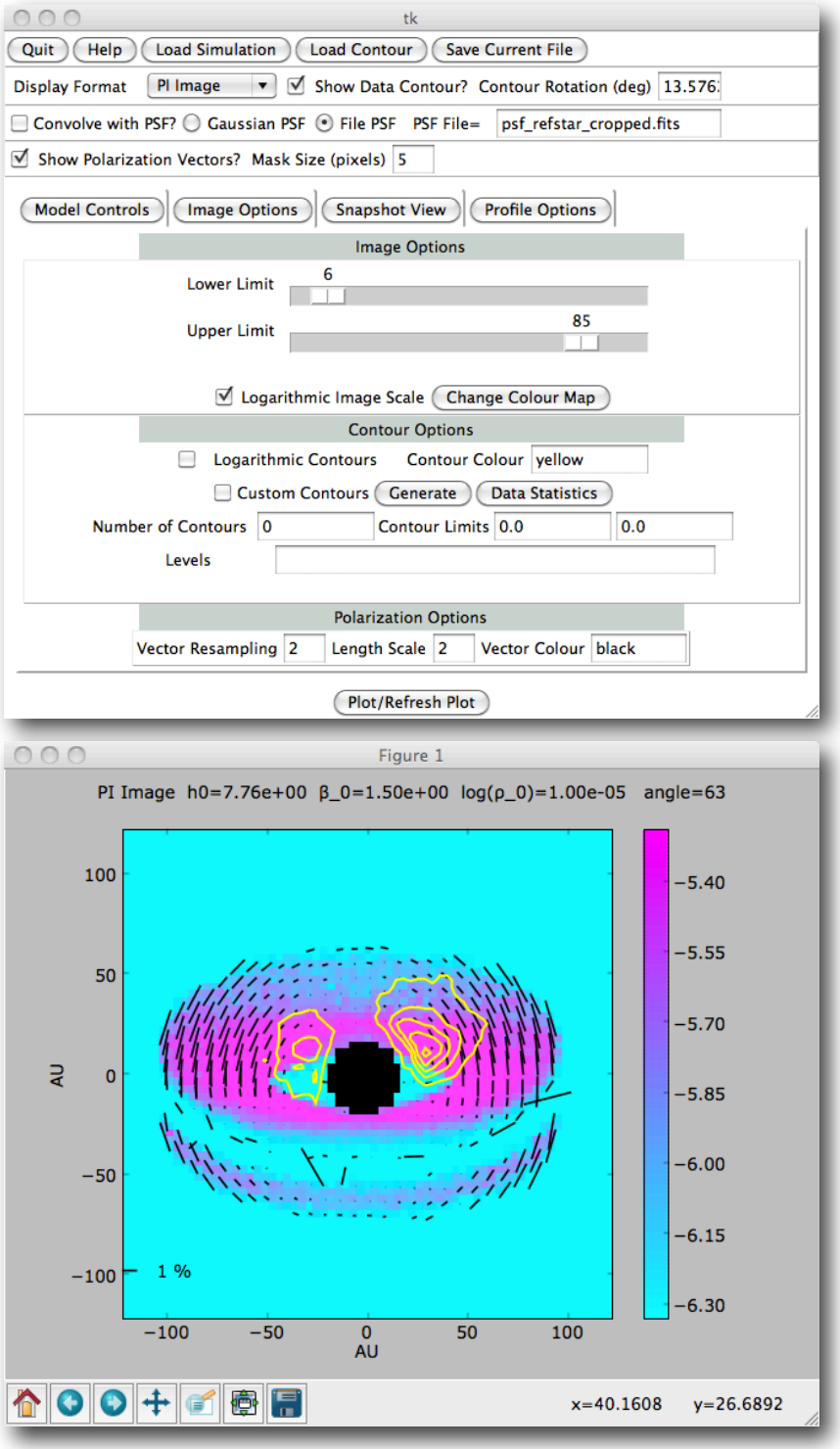


***Figure 12**: Customizing the Image - Settings for a highly customized view of the same model as Figure 10 (top) and the corresponding image (bottom). A PI image is shown (unconvolved) with data contours in yellow, rotated by 13 degrees. The color-map has been changed, and a logarithmic image stretch used, with the data cropped at the 6th and 85th percentile. The polarization vectors are shown in black, with the the length scale doubled, and resampled by a factor of 2.*

[Back to Text]

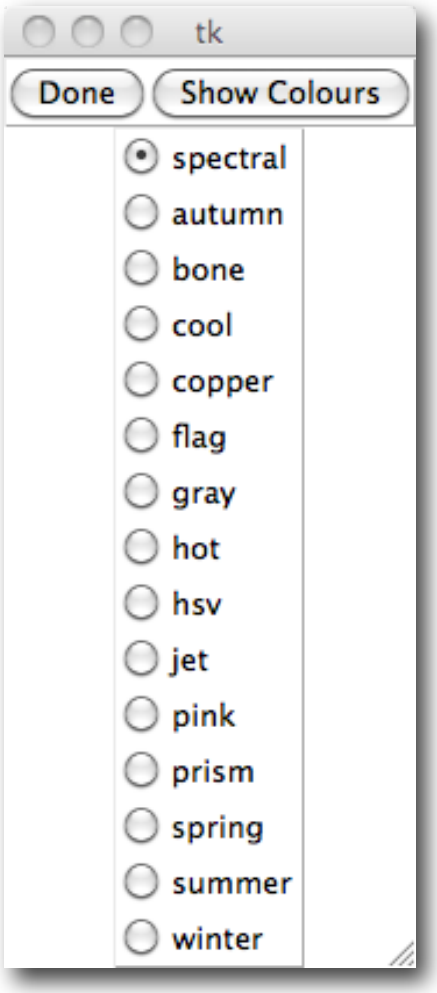
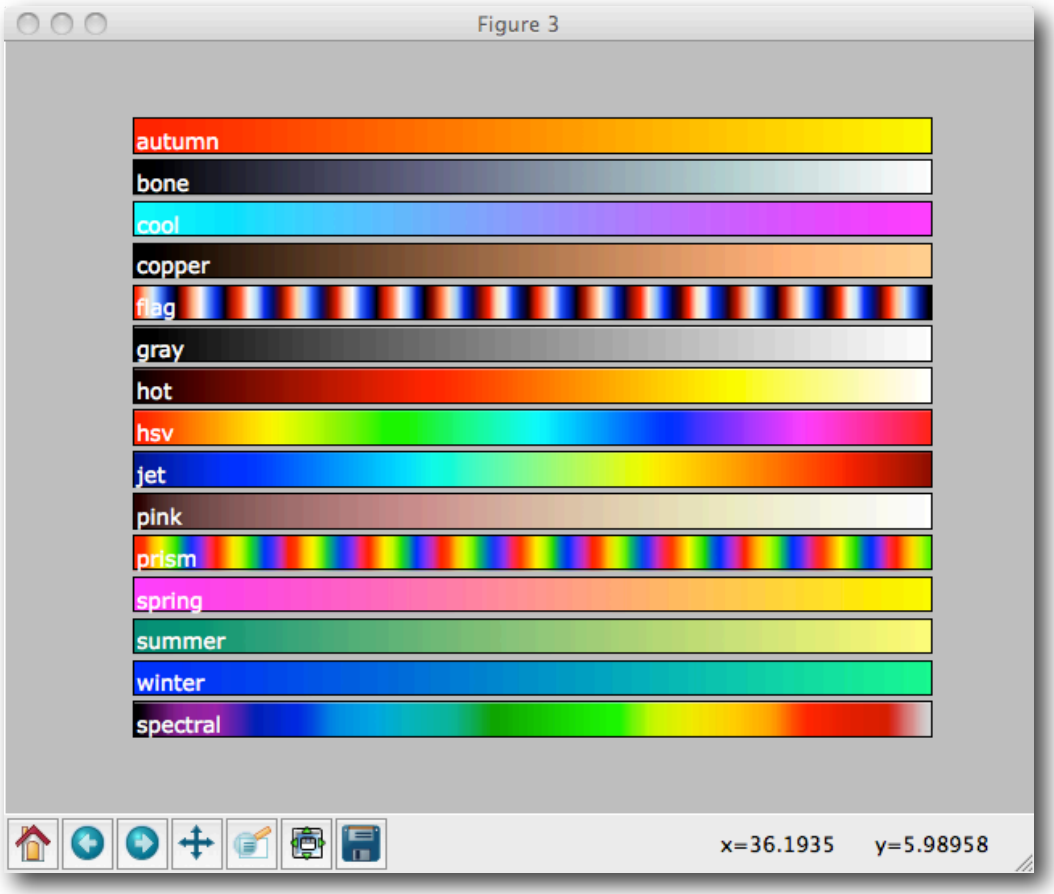
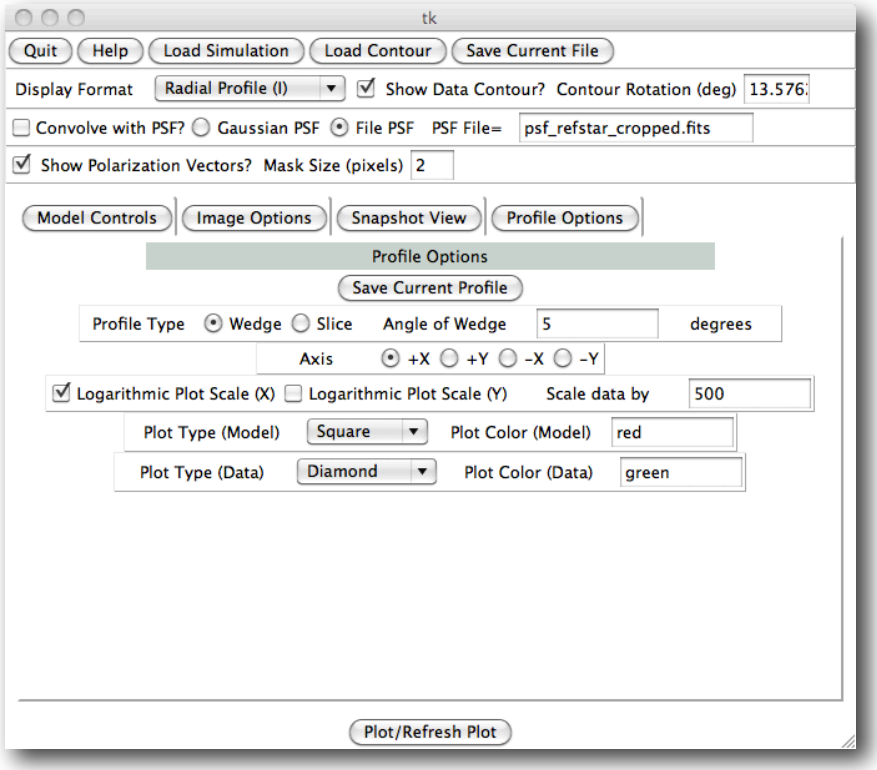
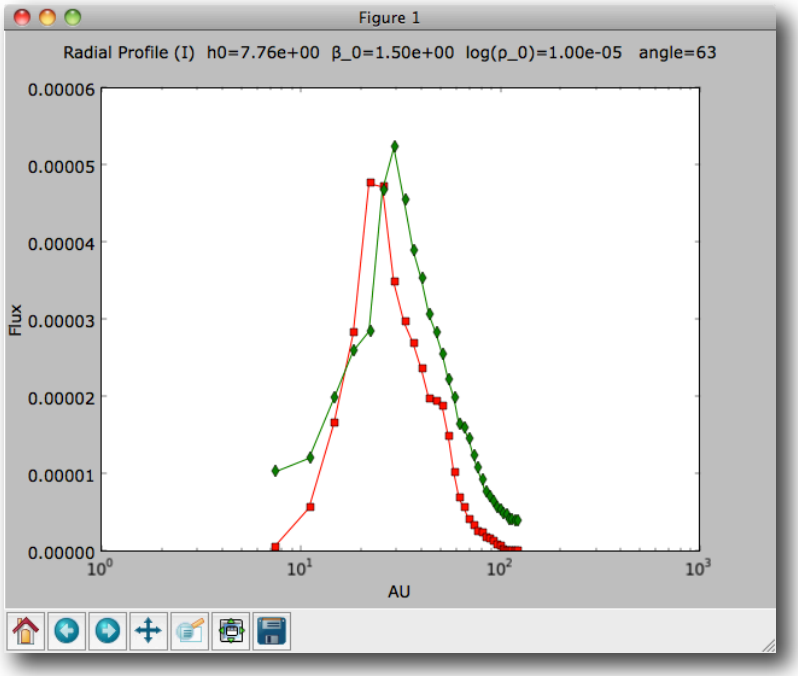


***Figure 13**: Changing colour maps. Left: selection of different colour maps. Bottom: illustration of the options.*

[Back to Text]







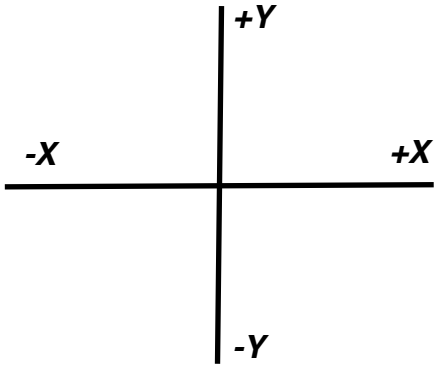


Figure 14: *Top: The Profile Options tab. Bottom Left: Radial profile plots for the model of Figure 10, showing the data values (green triangles) and the mode values (red squares), logarithmically in X and linearly in Y. The profile has been calculated along a wedge 5 degrees in width, along the positive X axis. Bottom Right: Definitions for the axis choices.*

[Back to Text]

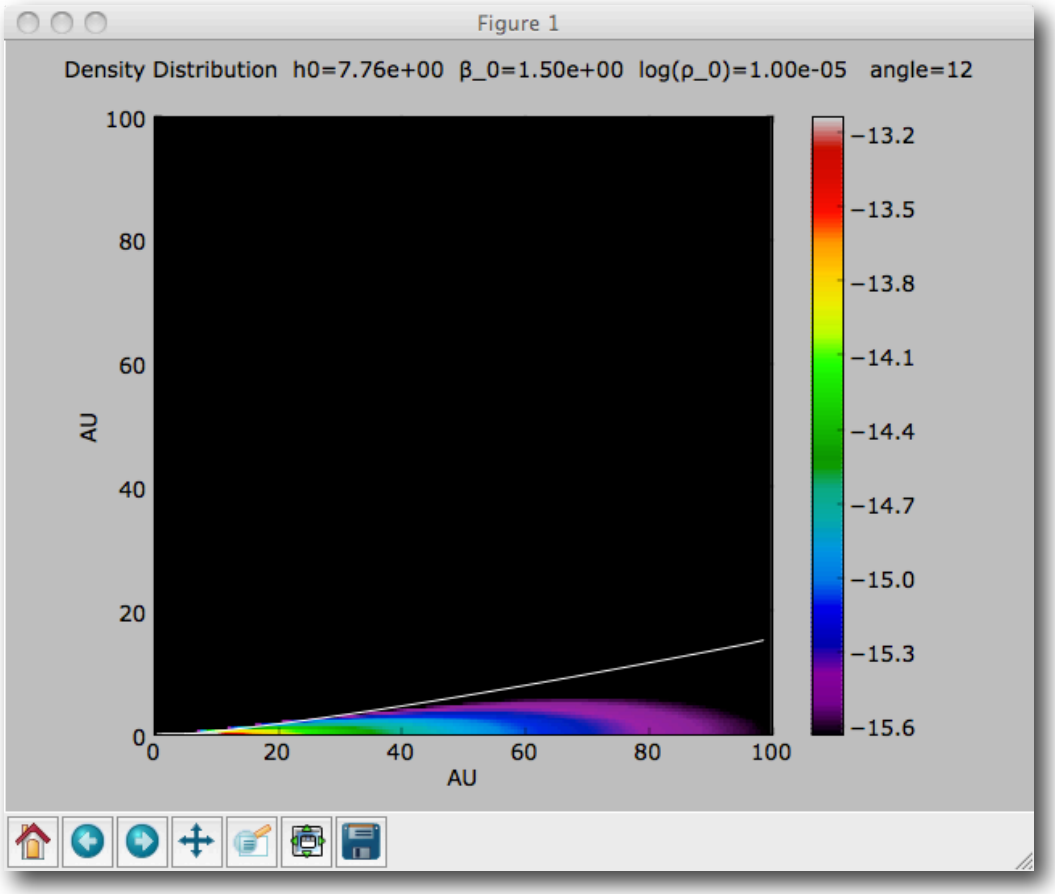


Figure 15: Plotting the Underlying Density Distribution. An edge on view of the density of the disk, shown in a logarithmic scale. The white line indicates the locus where the optical depth τ=1, measured in a line from the star.

Return to Text

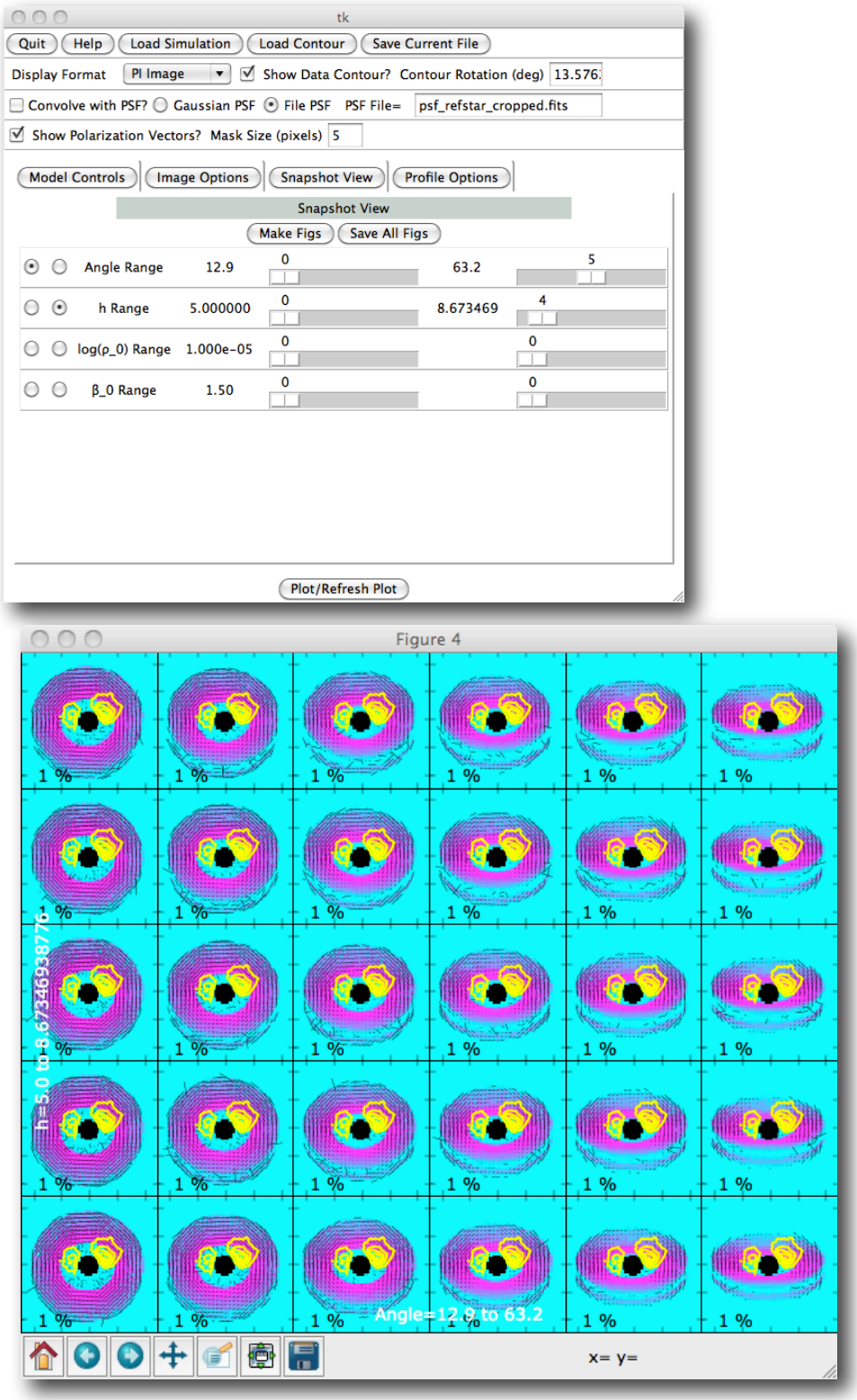


***Figure 16**: Top: The Snapshot View tab. Bottom: An image generated using the viewing parameters of Figure 10. In this case, values of h_o from 5 to 8.7 are shown (vertical axis) as well as viewing angles from 12.9 to 63.2 degrees.*

[Back to Text]