Shape

User Manual
Version 1.0

Autors:

Wolfgang Steffen
Instituto de Astronomía
Universidad Nacional Autónoma de México
Sede Ensenada, Baja California, México
wsteffen@astrosen.unam.mx

Nicholas Koning
University of Calgary
Calgary, Canada

CONTENTS

<u>OVERVIEW</u>

What is *Shape*?

Shape is a morpho-kinematic modeling tool for astrophysical nebulae. The main application of Shape will be the direct comparison of a 3D model with data in order to assist in the interpretation of observations. Starting from a 3D model of points with velocities, it generates an image and position-velocity (pv-) diagram, channel map and one dimensional spectral line shapes based on the Doppler effect.

What can *Shape* do for you?

The design purpose of Shape is the analysis of the 3D structure and kinematics of spatially resolved astrophysical nebulae in a way that can be compared directly with observations of similar real objects. It is particularly suited to study expanding nebulae like planetary nebulae and other structures with clear kinematical signatures like bow-shocks or accretion disks and other streaming flows that can be studied using the Doppler-Effect.

The analysis may be applied for the interpretation of existing observations or the planning and prediction of observations based on a proposed model.

Another potential application is the modeling of complex 3D structures as initial conditions of numerical photo-ionization and hydrodynamic calculations as well as the visualization and calculation of spectral information resulting from such calculations.

The application of *Shape* as a teaching and learning tool for the interpretation of observations involving images, position-velocity diagrams, channel maps and spectral line profiles may help students and researchers alike become acquainted quickly with this type of data.

What does *Shape* *not* do?

Shape does not calculate the numerical hydrodynamic evolution nor radiation transfer for photo-ionization models. *Shape* does not take a set of observational data as input to automatically yield a three-dimensional model.

Non-astrophysical uses

Shape may be applied to the analysis and visualization of data and models of objects observed with Doppler radars, such as tornados and other weather phenomena. Ocean surface observations with Doppler radar can also be modeled with some 3D animation packages. The application to the analysis of other types of Doppler data may, of course, be also possible.

Have you got news for us? User publications and utilities page

A webpage for *Shape* displayed at startup in the main interface of the software is a direct window that takes you right to the latest info on applications of *Shape* in the scientific literature. The page provides access to important up-to-date information on the software as well as model templates and model data from real applications that have been supplied by their authors. You are welcome

to make comments and suggestions for future releases of *Shape* and to share your own results with other users. Of course, as soon as a significant number of users is established, a discussion group will allow quick exchange of questions and answers to news and problems with model building in *Shape.*

TO TAKE INTO ACCOUNT BEFORE USING *SHAPE*

Citations:

In publications that include models that have been prepared with *Shape,* credit should be given citing the following original publication: Steffen, W., López, J.A., 2006, *Morpho-kinematic modeling of gaseous nebulae with SHAPE,* RevMexAA 42, 99-105.

Disclaimer

This software is provided "as is". No warranty is given for its fitness for a particular purpose. Neither the authors nor their employers can be held responsible in any way for incorrect scientific or other results that may follow from the usage and interpretation of models produced with *Shape.*

General modeling strategy

Given a set of one or more spectral line images and position-velocity diagrams or channel maps, a first interpretation of the data should give an approximate "educated guess" of the 3D structure and kinematics of the object. This structure is then reproduced in a suitable 3D modeling program like Blender, 3DStudioMax, Maya, or the built-in 3D module of *Shape*. You export the vertices or particles of the structure built in an external program, including their velocities, in a format that is readable to *Shape* (see below for how that may be done). From these data *Shape* then renders images, pv-diagrams and channel maps for different observational settings like object orientation, slit positions and slit widths, as well as different spatial and velocity resolutions.

The parameters settings for the model and observing setup may be varied automatically to produce a series of different image sets that you can view in an animated form. Viewing animations of image sequences often aids at understanding the effect of the 3D structure and orientation on the spectral information. If the model does not reproduce the observations satisfactorily in any orientation or with any other adjustment that can be made in the main interface, you go back and refine your model taking into account the differences between model and observation. Iteratively you will improve your model until you are satisfied. The process may also reach a limit when you hit the limits of your modeling software or when the assumption made in the models and their rendering do not apply to the real object (for instance optical thickness).

After arriving at a satisfactory model solution, the result may be inspected in various ways and scientific conclusions regarding the object structure, kinematics and orientation may be drawn.

In most cases observational data of an astrophysical nebula are not sufficient to derive a unique solution for its 3D structure and kinematics. However, in many such cases, modeling with *Shape* can still help to gain new insight into the structure of the object. Objects that have relatively simple topology or some noticeable symmetry can usually be solved uniquely. It is recommendable to exploit any hint to symmetries or surface continuities, both, in the images as well as the spectral information even if such properties can be found only in local feature of a complex object.

Knowledge required prior to using *Shape*

In order to profitably use *Shape* you should have at least basic knowledge about astronomical observing techniques like imaging, spectral line profiles, position-velocity diagrams and/or channel maps. The meaning of seeing and spectral resolution should be known.

Although desirable, for simple modeling using the 3D module of *Shape* no previous knowledge about 3D modeling is required, since it can be based on the templates available from the repository on the *Shape* website. For complex modeling of objects that can not be obtained by simple modifications of the templates, you can produce your model using a 3D software that is capable of exporting your model in the VRML (Virtual Reality Markup Language) format, some other format that is available or in the native format of *Shape* via a script of the modeling software. *Shape* provides data export scripts for some 3D modeling suites (see below for details). Check the *ShapeSite* (http://www.astrosen.unam.mx/shape) for currently available

Software installation

See the download section of the *ShapeSite* for details.

The rendering engine of *Shape*

The rendering of images and spectral information is an integrated projection of emission from individual particles (or polygon vertices which are treated as particles) onto the image or spectral plane. Color and brightness of the particles are controlled via a special color editor module. The individual particles are treated as circular planes which always face the camera with adjustable size and internal emissivity distribution.

Limitations

In Version 1.0 of *Shape* the integration along the line of sight is assumed to be optically thin. Capability to model optically thick nebulae will be included in Version 2.

Considerations on image resolution in particle based models

In order to obtain models with a complex structure, the number of particles that samples these structures has to be sufficiently high. The size of the convolution with the gaussian point-spread function should be sampled with at least a few particles. Note that structures that might not be resolved in the image, might well be resolved in the spectral information, such that a similar criterion has to be applied to structures that have details in velocity space that are "larger" than the velocity resolution.

USER INTERFACE

The user interface has two main modules: the Main Control Interface (MCI) and the 3D module.

When you run *Shape*, the Main Control Interface opens. Initially the MCI shows the menu bar and a row of tool buttons (*Button Toolbar*) the large News Panel, which gives access to up-to-date information on *Shape*. The MCI can be resized to suit your working area on the computer monitor maintaining access to the complete interface using the scroll bar on the right.

In order to start using *Shape* for modeling and visualization you open a project or create a new one using the *File* rollout on the menu bar at the top. Now the News Panel disappears to make way for the image displays and the parameter control tabs. The top displays show the raw image (left) and the raw pv-diagram or channel map (right). Similarly, the lower displays show image information on the left hand side and spectral information on the right. In the bottom displays the images have been convolved with gaussian point-spread functions like seeing and velocity resolution.

The middle section contains the control parameters for the rendering of the images and spectral information (see below for details). This section includes parameters for the space that is covered by the image, the velocity range that is covered by the spectral information, as well as controls for the automatic rendering of sequences with varying parameters.
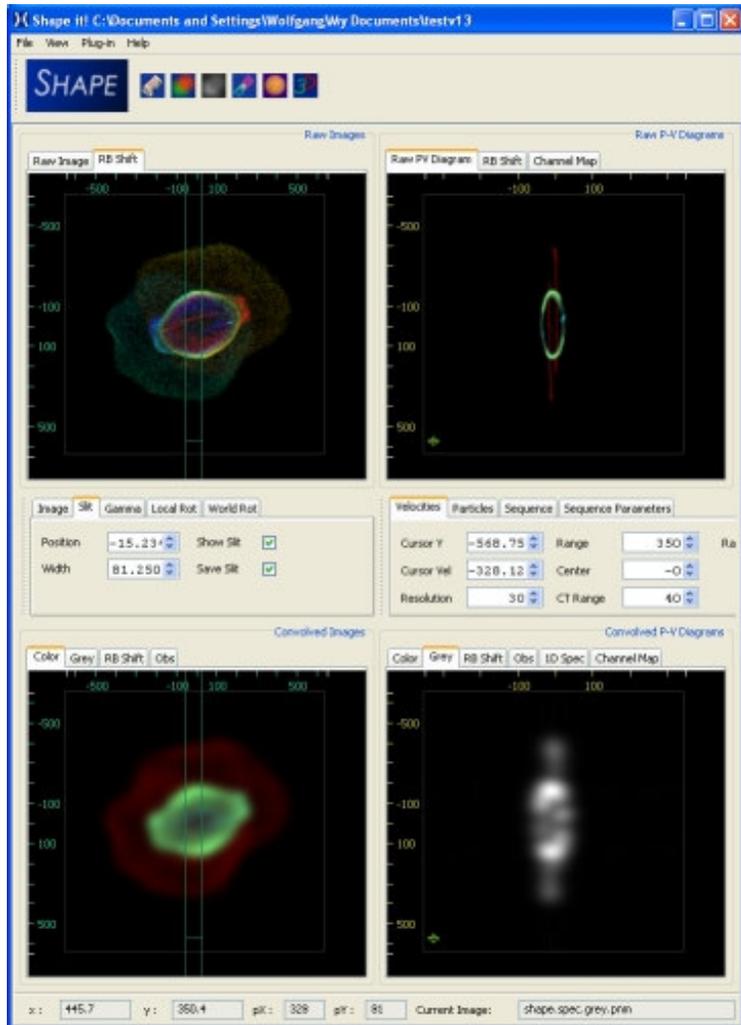
The news panel

Is your window to the **ShapeSite**, the web site with news and up-to-date information about *Shape,* including models, templates, tutorial and news about applications of the user community of *Shape.* The *News Panel* is displayed when you start a session with *Shape.* Once you start working on a project, the *News Panel* disappears. You can toggle between the *Project Panel* and the *News Panel* any time by pressing the *News Panel Button* on the tool bar at the top of the *Main Control Interface* (see below).

The *News Panel* at this stage is a limited *Browser.* You can only browse forward. To get back to the home page, press the *Refresh* button. At this time you can not download files using this page. To download templates and movies use the Internet Browser installed on your system pointed at the following Internet address:

http://www.astrosen.unam.mx/shape

Use the *News Panel* to learn about updates and news as well as to read the *User Manual.*
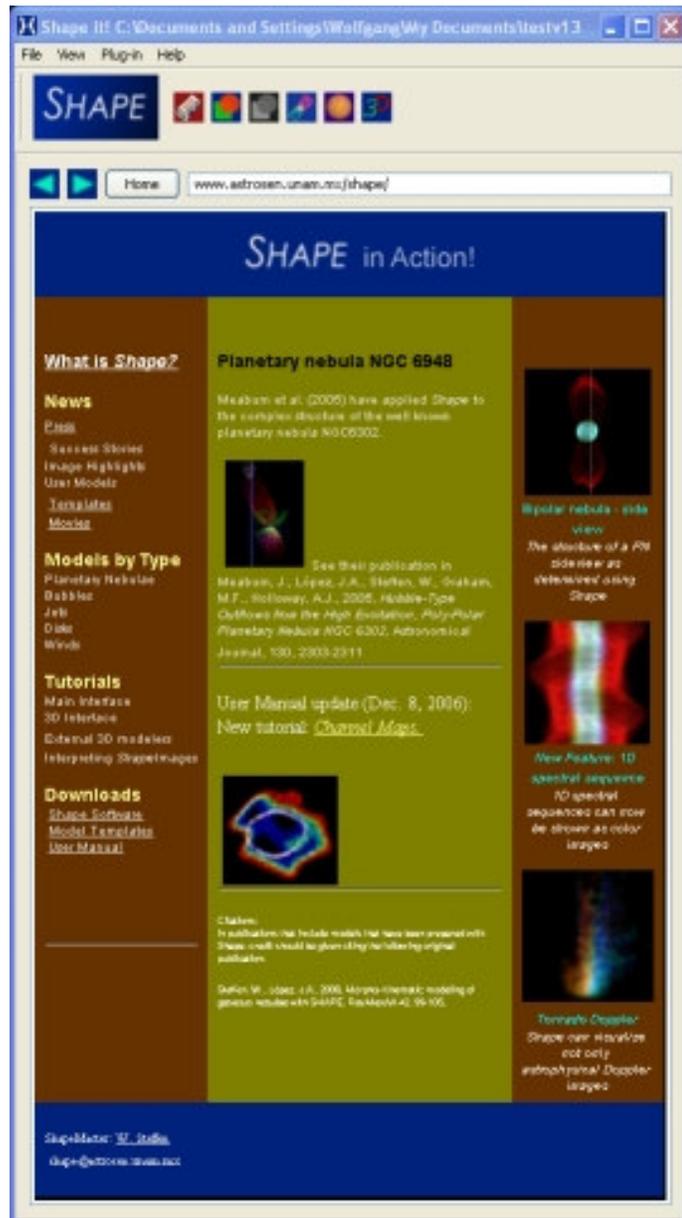
THE MAIN CONTROL INTERFACE (MCI)

WORKING WITH PROJECTS, DATA AND PARAMETER FILES

File   View   Options   Plug-in   Help   Menu Bar

Opening an existing project:

To open a recently used project open the ***Open Recent Project*** selector in the ***File*** rollout and choose one of the four most recent projects that you have worked on. The most recent is at the bottom of the list.

Menu Bar > File > Open Recent Project

To access other existing projects use the ***Open Project*** Dialog in *File* rollout of the ***Menu Bar***.

Menu Bar > File > Open Project

Creating a new project:

In order to create a new project with and existing Particle System data set open the ***Create New Project*** Dialog opening the *File* rollout in the *Menu Bar* :

Menu Bar > File > New Project

The ***Create New Project*** Dialog opens. In the first text field you enter a ***Project Name***. In the bottom text field you enter the ***Project Directory*** path, which will be your ***Working Directory***. The full directory path is either entered through keyboard input or by selecting the directory with the *Browse* to the right of the *Directory* text field. All the data that are created for this project are stored in the *Working Directory* or in a sub-directory of it. After confirming the operation pressing the *OK* button, the *Working Directory* is created in the *Directory* path. In the *Working Directory* a file is created with its name equal to the *Project Name* and the extension *".shp"*. In order to start working with an existing particle system data set, you import your correctly formatted file to the project using the *File* rollout

File > Import > Data.

A file selection window allows you to select the data file that you wish to import. After importing your data save the project (***File > Save Project***). Saving the project will include the data in the project file, including color and brightness of the particle systems in the data file. In addition, the file "***shape j3d.dat***" will be created, which is an identical copy of your original data file. The file *shape j3d.dat* will be used by the rendering engine of *Shape* to generate images and spectra.

Export data and parameters:

If you wish to save your particle system data file and/or the parameters of the current project in a new file for later use or backup, you open a *Save* dialog following

Menu Bar > File > Export > Parameters

Menu Bar > File > Export > Data

according to the file that you will save. You then choose the path and file name in the *Save* dialog that opens and confirm the operation with the *Save* button of the dialog.

Close a project:

Menu Bar > File > Close "*Project name*",

where "*Project name*" is the name of the currently open project.

LOOK AND FEEL

Every operating environment has a different "**Look and Feel***" (**L&F**). Although an interface may have the same features and structure, the way in which windows, buttons, tabs, etc. are shown on screen depend on the operating environment that you are using. The MCI of *Shape* is optimized for the **Windows XP** environment. Some people might however, be used to a different environment and prefer to change the *Look and Feel* of the MCI. You can do this by choosing one of the items in the selector following:

Menu Bar > View > Look and Feel

Macintosh L&F is not available on Windows or **Linux** systems. It is, however, the default on a *Macintosh* system.

PLUG-INS

The *Plug-in Menu Bar* item may contain a selection of modules that operate in windows that are separate from the MCI. In *Version 1.0* of *Shape* the only plug-in that is available is the *3D Module* (see below). Selecting the *Shape3D* item of the *Plug-in* rollout opens the *3D Module.* This selection has the same effect as pressing the *3D Button* on the *Button Toolbar* (see below).

HELP

The *Help* item on the *Menu Bar* gives access to information about *Shape.* Selecting the item *Contents* opens this **User Manual** in your default *web browser*.

Menu Bar > Help > Contents

For quick access to versioning, authoring and copyright information of *Shape* choose the *About* item of the *Help* rollout.

Menu Bar > Help > About

BUTTON TOOL BAR

SHAPE Button

 The *SHAPE Button* is the large button labeled *SHAPE* on the left side of the **Button Toolbar**. You press this button to invoke the rendering of images and spectra with the *Shape Rendering Engine.* Doing this produces a series of events. These events include saving the current parameters of the MCI, saving the parameters and data from the *3D Module* (if open), running the *Rendering Engine* and displaying the results in the MCI.

News Panel Button

 The *News Panel Button* toggles between the *News Panel* (see above) and the modeling panel.

Convolve Color Images Button

 Selecting the *Convolve Color Images Button* tells the *Shape Rendering Engine* to render the convolved color images and spectra. It does also control which images are replayed during replay of a sequence using the corresponding features in the *Sequence Tab* (see *Sequence Tab* below).

Convolve Grey Scale Images Button

 Selecting the *Convolve Grey Images Button* tells the *Shape Rendering Engine* to render the convolved grey-scale images and spectra.

Model / Observation Button

 The *Model / Observation Button* toggles between the display of the convolved color image and spectrum and a corresponding observed image. This allows a comparison of observation and model by quickly "blinking" between them in the same display. The observed image should be in the working directory and have the name "shape.obs.image.jpg" and

12

"shape.obs.spec.jpg" for the image and spectrum, respectively. Note that the centering and extension of these images should match those of the parameters used in the displays (or vice-versa).

Opaque Star Button
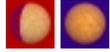
 The *Opaque Star Button* allows to render a rotating or expanding opaque star. When selected all particles on the far side of the star are set to zero brightness. This allows to simulate the spectral information of a rotating star with surface structure. Using a sequence of a rotation of the star produces a Doppler-tomogram of 1D Spectra in the corresponding image (see below).

3D Button

 Pressing the *3D Button* on the *Button Toolbar* opens the **3D Module**. The *3D Button* has the same function as the *Shape3D* item in the *Plug-in* rollout in the *Menu Bar.*

PARAMETER TABBED PANELS

Parameter values may either be entered in the corresponding text field using keyboard input or by pressing the up or down arrows of the "spinners" located to the right of the text fields. Briefly pressing the spinners will change the value by a given amount, which may be different for different parameters. Keeping the spinner pressed will continuously increase or decrease the number in the text field. Some parameters may also be changed by moving the mouse in the corresponding display.

Image Tabbed Panels

***Image Tab*:**
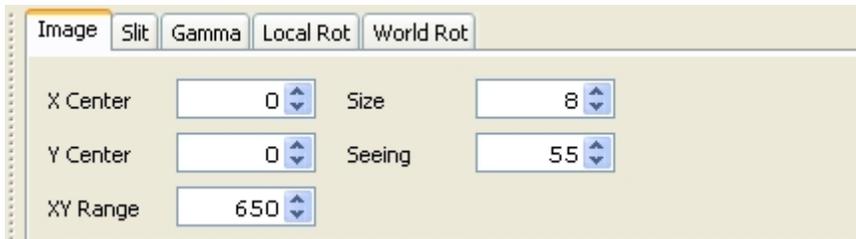
Controls the parameters of the window into the *World Coordinate System* that is seen in the image displays.

X Center, Y Center: The coordinates of the image center in the model world space. Horizontal tick- marks show the x-coordinate system. The y-coordinate values can be found using the cursor on the PV-diagram. Tick-marks, slit and cursor are only shown, if *Show Slit* in the *Slit Tab* is selected (see below).

XY Range: The size of the image from the center to each border.

Seeing: The Full-Width Half-Maximum (FWHM) of the gaussian convolution kernel for the simulation of the blurring effect ("*seeing*") of the terrestrial atmosphere on optical images of astronomical objects. If the *Convolve Images Buttons* (see above) are selected, the spatial dimensions of the images and spectra are convolved with this kernel.

Size: The pixel size of the rendered images is 2^{Size}. The images are then scaled to the 2^8 (256) pixel size of the displays in the MCI. Reducing the image size results in a faster computing time, which may be significant if convolution of the images is performed and resolution is not important, e.g. during testing and quick parameters searches. Larger images sizes to result in slower performance, but higher resolution.

Slit Tab:
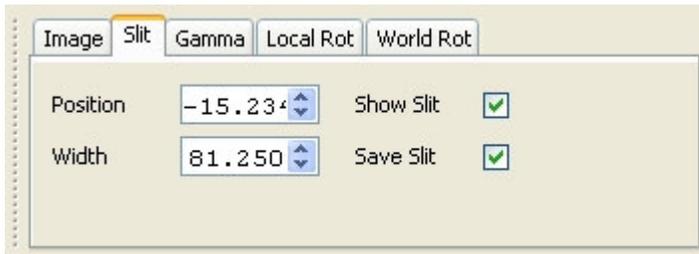
Controls the position and width of the simulated spectrograph slit in the world coordinate system.

Position: The position of the slit center along the x-coordinate of the world coordinate system. The values can either be introduced manually or my positioning and clicking the cursor on the image.

Width: The width of the slit center along the x-coordinate of the world coordinate system. The values can either be introduced manually or by scrolling the mouse-wheel if available.

Show Slit: Choose whether to show the image slit and the scales on the images and pv-diagrams. Deselecting this check-box is useful to improve performance and visibility while replaying image sequences.

Save Slit: Choose whether to show the image slit on the rendered images that have been saved to disk.

Local Rot **and** ***World Rot Tabs***

These tabs control the orientation of the object with respect to the observer in the *Local and the World Coordinate System.* Values given are rotations in Degrees around the corresponding axis.

x-axis: horizontal increasing towards the right
y-axis: vertical increasing upwards
z-axis: perpendicular to the screen, increasing into the screen

With all the parameters at "0", both coordinate system are identical. However, if the the parameters of the rotation in the *World System* are changed, the whole *Local System* rotates. Changes to the *Local System* are applied after the *World System.* Note that rotations can not be performed in arbitrary order.

Gamma Tab

**The *Gamma Tab* controls the *Gamma Correction* value applied to the images.**

The *Gamma Correction* performs the operation **pixelvalue^(1/*Gamma*)**, where *pixelvalue* is the intensity of the any picture element. Increasing *Gamma* values enhances the relative brightness of low surface brightness features.

Image Gamma: Gamma value applied to all image displays
Spectrum Gamma: Gamma value applied to all spectral displays

Spectral Tabbed Panels

Controls the parameters of the spatial and radial velocity coordinates that are seen in the spectral displays.

Velocities Tab
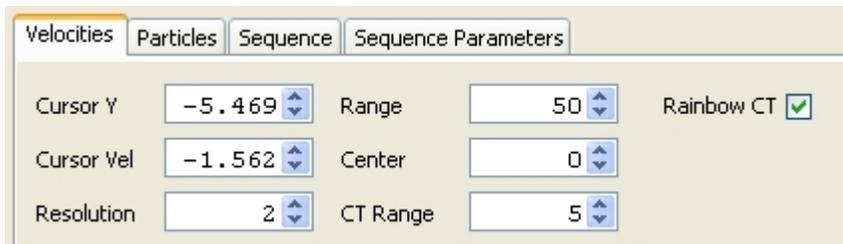
Center: The velocity at the center of any P-V diagram. Velocity is horizontal in the P-V diagrams
Range: The range of velocity from the center of any P-V diagram to the left and right edges
Resol: The FWHM of the gaussian velocity convolution kernel. This convolution simulates the velocity resolution due to natural and instrumental effects.

Cur Y: The y-position of the cursor in the p-v diagram in world coordinates.
**Cur Vel:** The velocity at the position of the cursor in a display with horizontal velocity axis (P-V diagram or 1-D spectral line profile).

Rainbow CT: Selecting this checkbox will enable the *Shape Renderer* to apply a *Rainbow Color Table* to the radial velocity color coding in the P-V diagrams, instead of a simple red-blue coding which only shows whether a particle is approaching of receding. The rainbow color table allows a differentiation of the actual radial velocity.

CT Range: The range of radial velocities around the *Center Velocity* that will be coded with the *Rainbow CT*. All velocities outside this range will be grey, with values of 0.3 times the original, in order not to be outshining the color coded regions. This functions helps to visually extract regions of a particular range of radial velocities in the images.

Particles Tab

These parameters control the size and internal brightness distribution of the particles in the raw images.
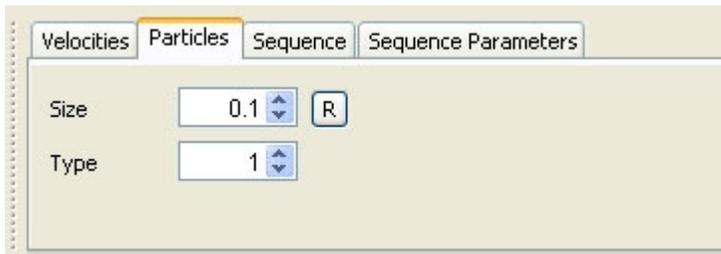
Size: The radius of a particle in units of the *World Coordinate System.*

Type: Currently only one internal brightness distribution is implemented. It is a bright central spot with a weaker halo reaching out to the *Size* from the center of the particle. Note that in the convolved images and spectra the internal structure of the particles is erased, since the convolution kernel is (should be) larger than the size of a single particle.

R-Button: Restores the *default size* for the particles, which is 10 percent of the *xy-Range* of the *Image*.

Sequence Tab

The *Sequence Tab* controls the calculations of a sequence of images and spectra varying one or more parameters of the MCI.
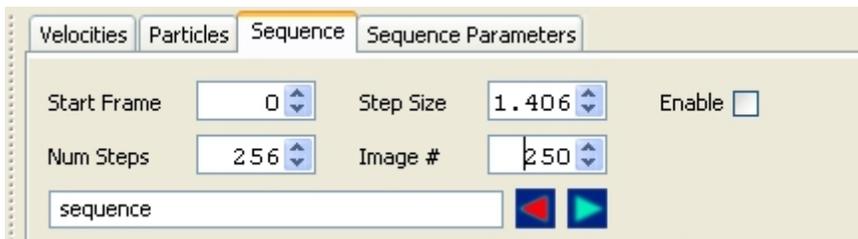
The parameters to vary are chosen in the *Seq Par Tab* (see below). The resulting images are written into a sub-directory of the *Working Directory*. The name of the sub-directory is given in the text field at the bottom of the *Sequence Tab.* Each image of the sequence has the sequence name and an *image number* included in its filename.

Enable Checkbox: Enables the rendering of an image sequence

Start Value: This value is added initially to all the parameters that vary in the sequence. The final start value is the sum of this value and that shown by the particular parameter text field.

Num Steps: The sequence will have this number of steps.

Step Size: The amount by which each parameter will be increased in every step. The value may be negative, such the parameter decreases during the sequence.

Image #: The current image number that was rendered or that is being displayed at playback of the sequence.

Sequence name: The *text field* at the bottom of the *Sequence Parameter Tab* contains the name of the sequence. When you start rendering a sub-directory will be created with this name. This name and an *image number* will be included in every image frame that is written in this directory.

Playback control: Press the left and right arrow icons to set the playback direction of the sequence. The sequence can be played back either by repeatedly pressing one of the playback icons or by pressing any keyboard key. Keeping the keyboard key pressed will automatically advance the playback. However, depending on the computing and disc reading speed, playback may not be continuous, but jump more than one image.

Seq Par Tab

In the *Seq Par Tab* you select which parameters will be varied in a sequence.
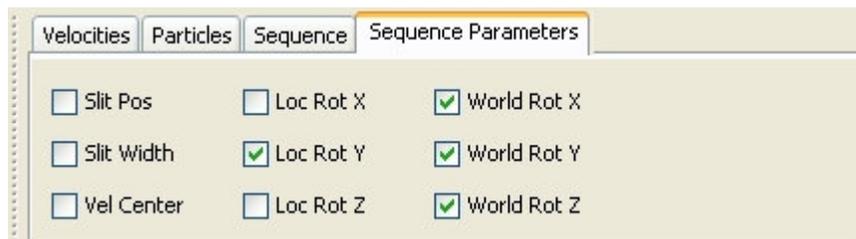
One or more variables may be varied simultaneously. The step size of the variation is the same for all. The initial value may be different for each of them and is given by the value displayed in the text field of the corresponding parameter plus the **Start Value** in the **Sequence Tab** (see above).

IMAGE DISPLAYS

Spatial projections of the model object and observations are shown in the displays on the left hand side of the MCI. Invisible images are accessed by clicking on the corresponding tab.

The image displays by default show the simulated spectrograph slit as a pair of lines. In the spectral renderings, only particles which are located within the slit are processed. You change the position of the slit using the mouse on the display or by typing the corresponding numbers in the *Slit Tab.* Similarly, the width of the slit is changed by scrolling the mouse wheel or typing the numbers in the corresponding text field in the *Slit Tab*. The horizontal line that connects the edges of the slit, indicates the vertical position of the cursor in the P-V diagram on the right to the image. This helps to locate corresponding features in the different displays. The plane of the image is given by the "x" and "y" coordinates of the *World Coordinate System* in the horizontal and vertical direction, respectively. The "z" coordinate is perpendicular to the image and increases into the image. Tick marks at the top of the display show the x-coordinates in the *World Coordinate System.*

Note: Since the rendering of the images is not a real-time process, once you change the *Parameters,* the images do not correspond to this parameter set until you re-run the rendering clicking on the *Shape Button* in the *Tool Bar.*

SPECTRAL DISPLAYS

Spectral information is shown in the right hand side displays. Invisible spectra are accessed by clicking on the corresponding tab.

The horizontal axis in these images measures the radial Doppler velocity as shown by the scale at the top of the display. The vertical axis is the position along the simulated spectrograph slit as shown in the image displays on the left. Only in the *1DSpec Tab* the vertical axis is not space, but the sequence number. A mouse cursor shows the current y-position and velocity. The elliptic part of the cursor is a visual cue for the size and shape of the convolution kernel applied to the convolved images. The horizontal width of the ellipse is the same as velocity resolution *(Resol)* in the *Velocities Tab.* The vertical size of the ellipse corresponds to the *Seeing* kernel applied to the images. For the scale and the cursor to be visible the *Show Slit Checkbox* in the *Slit Tab* must be checked.

Raw Images and Spectra

Raw images and spectra show the rendered object unconvolved with seeing or velocity resolution kernels.
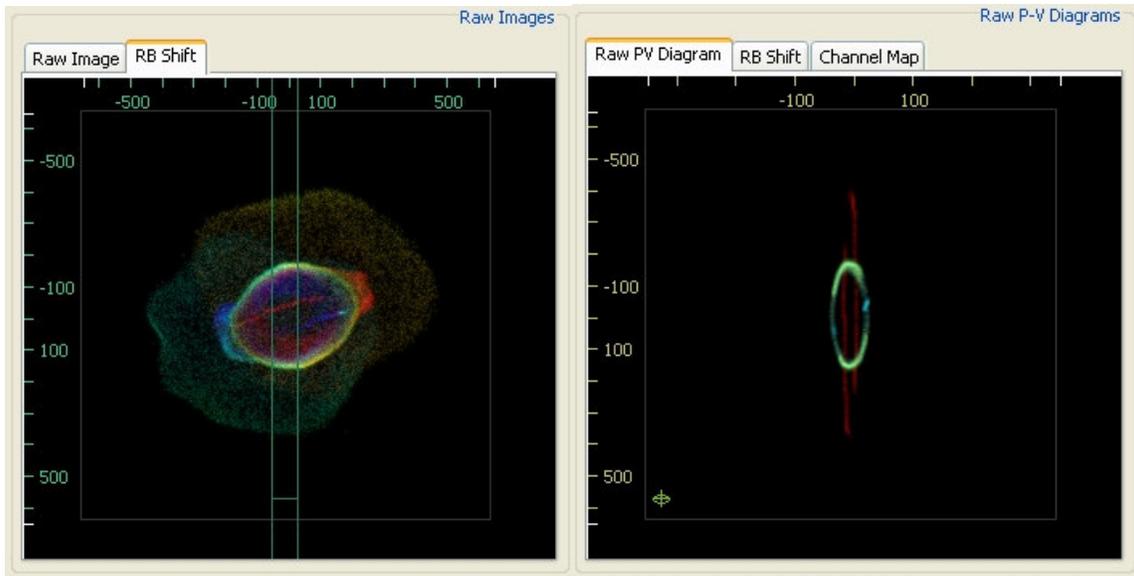
They are shown in the top displays. If the number and size of the particles is not too large, the individual particles can be seen in these images.

Convolved Images

Images in this display have been convolved with a circular gaussian kernel of FWHM equal to the *Seeing* Parameter in the *Image* Parameters Tab.
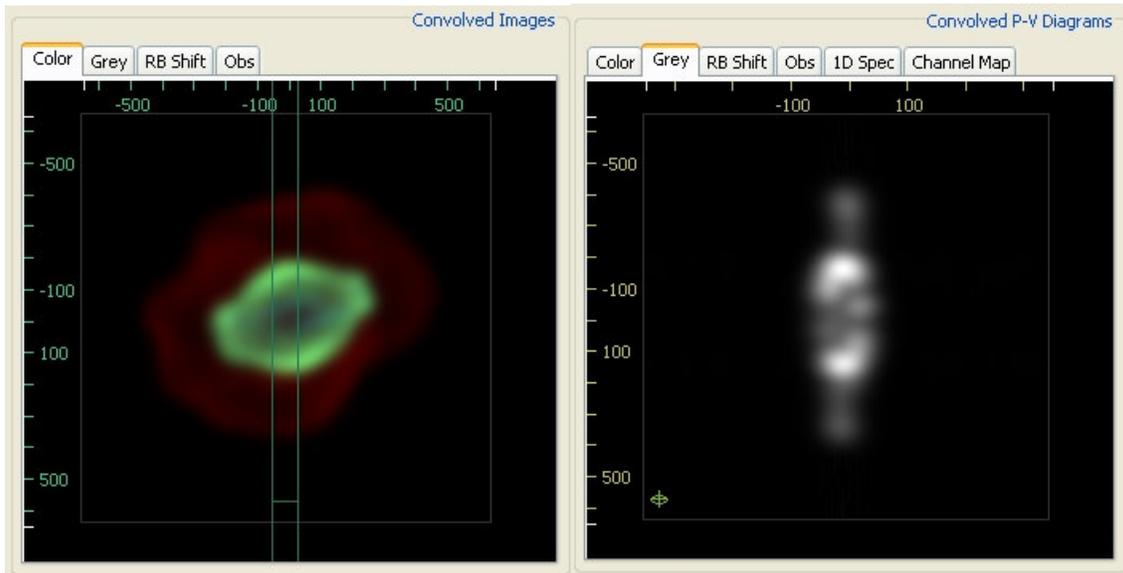
Color: Image with color coding according to the colors given to individual sub-objects. You can change the colors of the sub-objects from within the *3D Module*. The *3D Module* opens by pressing the *3D Button* in the *BUTTON TOOL BAR*.

Obs: Observed image to be used as a reference during modeling. You prepare this image in such that the orientation of the spectrograph slit used for acquisition of the P-V diagram on the right display is vertical. The file of the observed image is *"shape.obs.image.jpg"* and is located in the *Working Directory*.

Grey: The grey-scale image. Otherwise the same as the color image. No distinction is made between sub-objects.

RB Shift: This image is color coded according to the Doppler-Shift of the particles in the model.

There are two different modes for the color coding. The red-blue shift (RB-mode) renders a particle red when it is red-shifted and blue when it is blue-shifted weighted by their brightness. If the Doppler-shift is zero, the color is white. Whenever there is a superposition of red and blue-shifted particles, the color is a mixture of red and blue, weighted by their corresponding brightness.

The second mode is enabled by selecting the *Rainbow CT Checkbox* in the *Velocities* Parameters Tab (see above). It color codes the Doppler-shift in the colors of the visible spectrum, from blue to red, with the intensity proportional to the brightness. This color table is applied to the velocity range given in the *CT Range* Parameter to either side of the velocity center set in the *Center* Parameter of the *Velocities* Parameters Tab. Velocities outside this range are rendered in a brightness grey-scale. The *Rainbow CT* image and the RB-coded image are not available simultaneously. In order to obtain the image of the color coding that is not visible, change the *Rainbow CT Checkbox* and press the *SHAPE Button* to render the model again.

Chan Map: In this display the channel map is shown. Only particles of the model are included that have radial velocities (Doppler-shifts) within the *Range* Parameter center around the *Center* Parameter in the *Velocities* Parameters Tab. These are the same particles that are color coded with the *Rainbow CT* feature.

Convolved P-V diagrams

**Color:** P-V diagram with color coding according to the colors given to individual sub-objects.

Obs: Observed P-V diagram to be used as a reference during modeling. The file of the observed image is *"shape.obs.spec.jpg"* and is located in the *Working Directory*.
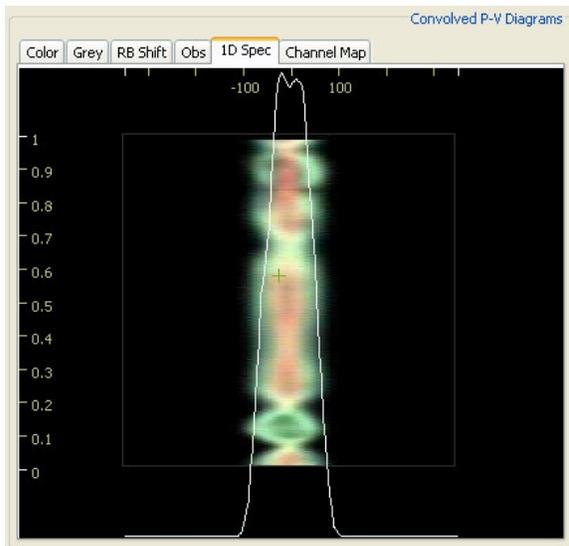
Grey: The grey-scale P-V diagram. Otherwise the same as the color P-V diagram. No distinction is made between sub-objects.
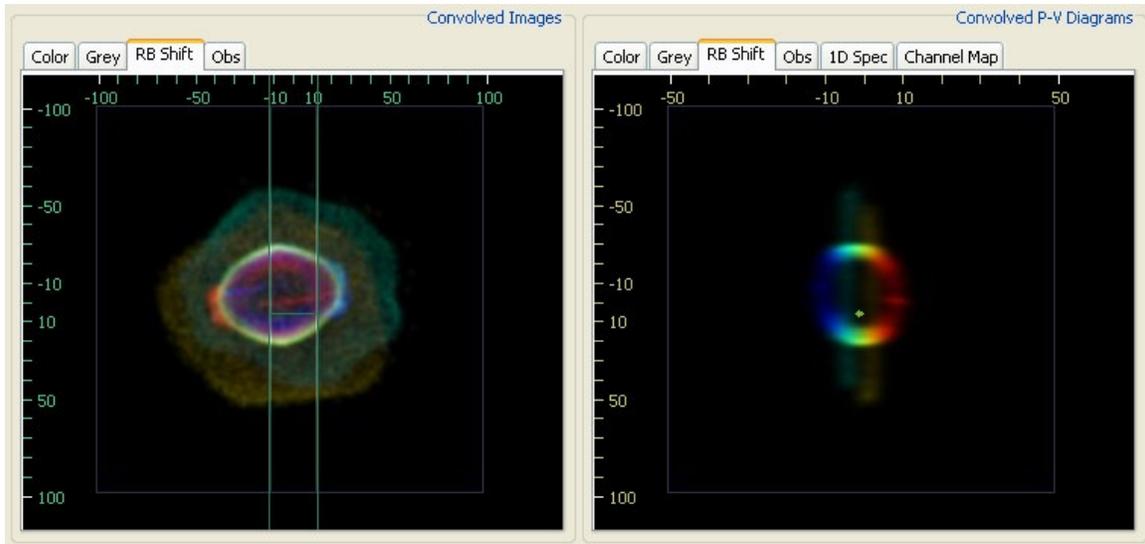
RB Shift:

This image is color coded according to the Doppler-Shift of the particles in the model. For further comments see the **RB Shift** section in the *Convolved Images* section above.

1DSpec:

In this tab the 1D spectrum is shown. It is normalized to the height of the display, i.e. the maximum peak is always at the top edge of the display. If a sequence was run, the sequence of 1D-spectra is shown as a color image. Each row of pixels contains the 1D spectrum with the intensity coding the value of the spectrum and the color codes the mixture of colors from the colors of the sub-objects. Colors that are not actually present in the model indicate a strong mixing of the sub-objects in the spectra. If the process that generated the sequence had the convolution enabled, then the line spectrum is also convolved. Otherwise, the raw spectrum is used. The velocity and the sequence number of a particular 1D Spectrum can be read by pointing with the cursor and reading the numbers in the *status bar* (see below).

Status Bar

| x :	445.7	y :	350.4	pX :	328	pY :	81	Current Image:	shape.spec.grey.pnm

The *status bar* at the bottom of the MCI provides information about the current cursor position on the images in *World Coordinates* and velocity (x, y) as well as pixel coordinates of the corresponding image (pX, pY). On the right side of the status bar the filename of the *Current Image* is given at which the cursor is pointing.

THE 3D MODULE

The *3D Module* has the function to control some properties of an object and its sub-sections. Among others, these properties include their position, size, orientation, color, brightness and, very importantly, their velocity structure. In this module, objects can be changed in their shape, like stretching them in a particular direction. For complex structural manipulation it is recommended to employ an external 3D modeling software and them import the data in a suitable format into *Shape*.
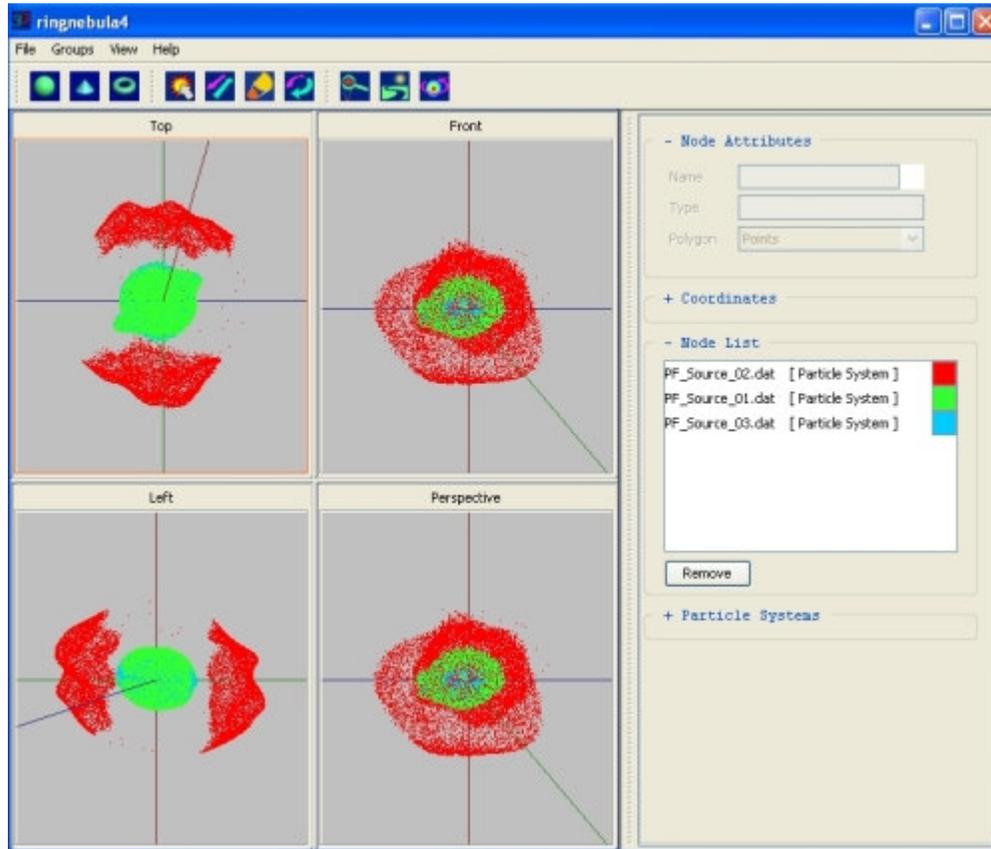
General strategy for the usage of the *3D module*

Any imported dataset is visible in the 3D module, situated in the *World* with its associated *World Coordinate System.* The data may either be in the form of a *Particle System (PS)* imported from the *shape j3d.dat* file or e.g. in the form of the *Virtual Reality Markup Language (VRML)* format. Some simple primitives like spheres and tori can be created directly from the *3D Module.* These primitives can be combined or deformed to yield disks, ellipsoids or other more complex structures. Any non-particle system objects (also called *nodes*) must then be converted to a particle system. After that you assign color, brightness and velocities. Orientation and any other transformations can either be applied before or after converting the *node* into a PS. Now you can render the complete object in the Main Control Interface (MCI).

WARNING: It is very much recommended to premeditate every operation that you will perform to an object. At this time there is NO GENERAL *UNDO* function available. All backward changes have to be done manually at this stage.

Menu Bar

File Groups View Help

File

Menu Bar > File > New

In order to start a new object from an empty *World* you select *New*. This will erase *all* previously existing sub-objects. *WARNING:* At this time this operation can not be undone directly and completely. There are three ways of recovering some or all of the information. If you have *saved the World* in this *File Menu* then you can *open* it again selecting **Menu Bar > File > Open.** Secondly, you can use the **File > Import > Data** in the MCI to reload the particle system data that have been saved when you last rendered the object. Third, you can reload the *project* from the MCI using **Menu Bar > File > Open Recent Project.**

Menu Bar > File > Open

Menu Bar > File > Save

Menu Bar > File > Import > VRML

Import a VRML object file. This will *add* the objects contained in the VRML file to the current *World.*

Menu Bar > File > Export > VRML

Export any VRML objects in the scene. VRML objects which have been converted to *Particle Systems* can not be exported in VRML format.

Menu Bar > File > Exit

Exit the 3D module.

Groups

Create and Remove *Group*

Create joins all selected sub-objects into a single group. Transformations like moving and scaling are performed to all sub-objects in the group simultaneously. *Remove* opens a group and separates sub-objects from a group.

Create and Remove *Particles System*

*Create*s a *Particle System* from the selected VRML objects. Destroys and *remove*s the object from the *world*.

View

Select which sections of the *Tool Bar* are displayed. The available tools are ***Primitives, Transforms, Camera*** and ***Node Information*** (see below)*.* The *Primitives, Transforms* and

24

Camera tool bar buttons are located above the ***View Panel*** where the objects are shown from different *Camera* directions. The *Node Information Panel* is located to the right of the *View Panel*.

Help

Opens this document.

Tool Bars

Primitives

Buttons to create new sub-objects in the form of simple geometric shapes *(primitives)*. The currently available *primitives* are *Sphere, Cone* and *Torus.*

To *create a new primitive* select the desired button and press the mouse in any of the *View Panels* (see below). Keep the mouse pressed and *drag* to the right and up to *increase the width and height*, respectively.

Transforms

Use this tool bar to ***Move, Scale*** and ***Rotate*** objects.

To apply any transform, you first select the object. Selection of an object can be achieved by either activating the ***Select Button*** and then clicking on some part of the object or by clicking on the object in the *Node List* of the *Node Information Panel.* To actually perform the *Transform* you activate the corresponding ***Transform Button*** *(Move, Scale* or *Rotate)* and finally move the object by dragging the mouse or introducing the transform values in the *Coordinates Panel* of the *Node Information Panel.*

To **restrict a transform** along a particular *axis* drag the mouse while pressing the *x, y* or *z* key of your keyboard. Choose the *Coordinate System* in which to perform the transform by selecting one of the radio buttons (*World, Local*) in the *Rotation Panel (Node Information Panel > Coordinates > Rotation)***.**
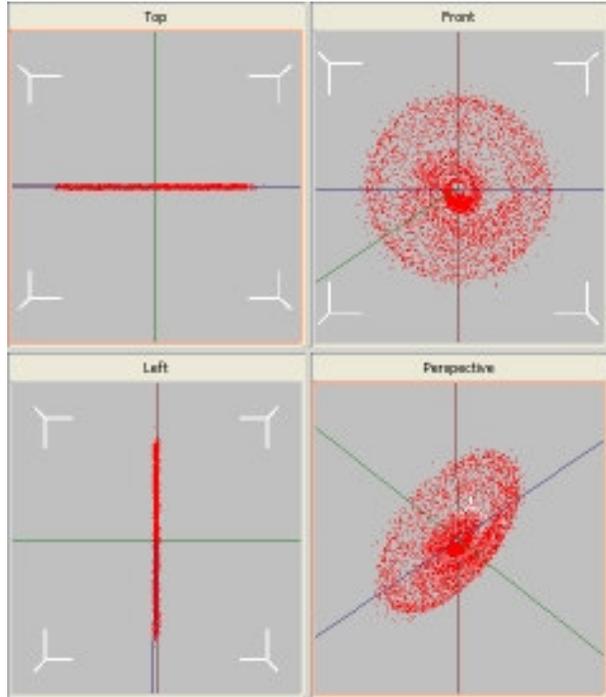
Camera

The *Camera Tool Bar* allows you to ***Zoom*** or ***Pan*** the viewpoint in all the *View Panels.* The *Perspective Panel* does also allow to ***Rotate*** the viewpoint around the origin of the *World Coordinate System.*

View Panels

The **View Panels** show the *World* from different directions, **Top, Front, Left**, **Perspective** and **Shape view**. The *Top* view looks along the *y-axis,* the *Front* view along the *z-axis* and the *Left* view is a projection of the *World* along the *x-axis.* At start-up the *Perspective* view looks along the z-axis and is the same as the *Front* view. In the *Shape view* the observer looks at the same scene as in the images in the MCI. It communicates directly with the values in the *Image Parameters Tab*.

Right-clicking with the cursor on the frame of a view panel opens a menu from which you can choose a different viewing direction. In some viewing directions, it does also allow to quickly adjust the size of the view panel to the size of the model objects. *Right-clicking* and dragging on the view panel directly invokes *panning*, i.e. moving the camera. Using the *mouse wheel* on the view panel allows to interactively *zoom* in or out.

Shape view: This view panel gives a preview for the rendered image of the MCI. It can be used to adjust the viewing direction applying the *Camera Transformations* of the *Tool Bar.* The values of the *Image Parameter Tab* are reproduced in the *Shape Camera Panel* of the *Nodes Panel.* They can be changed either in the MCI or in the *Shape View Panel.*

Nodes Panel

In the *Nodes Panel* you manage the model objects and it's parts. It is located to the right of the *View Panels.* There are four collapsible main panels: the *Node Attributes, Coordinates, Node List* and *Particle Systems.*

Node Attributes

The **Node Attributes** include the **Name, Type, Polygon** type and **Color**.

Name: displays the name of the node. It can be changed in the corresponding text field.

Type: shows whether the *node* is a *Particle System* or a *Polygon Surface*.

Polygon: select how a polygon surface is to be displayed, as points, wire frame or a shaded surface. This selection has no effect on *Particle Systems,* including surface objects which have been converted to *Particle Systems.*

Color: The color swatch shows the current color of the node. You can change the color by clicking on this swatch. The *color chooser* will open and allow you to set the color.
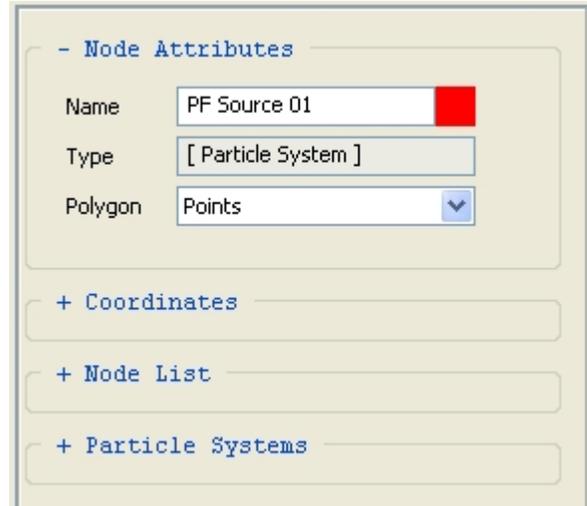
Coordinates

The *Coordinates Panel* holds the information on the *Position, Rotation* (orientation) and *Scaling* of the nodes.

Position

The *Position Panel* displays the *x,y* and *z* coordinates in the *World Coordinate System.* The values may be changed directly in the corresponding text fields. This allows precise placement of each node.

x, y, z: The *World Coordinates* of the *Node.*

Show Gizmo: A checkbox to show or hide the axes (**gizmo**) of the local coordinate system associated with the *Node.*

Scale: The scaling of the *Gizmo.*

Rotation

The *Rotation* determines the orientation of the object with respect to the various *Coordinate Systems* and hence with respect to the observer. The *Rotation* can be changed with respect to two *Coordinate Systems*, the *World*
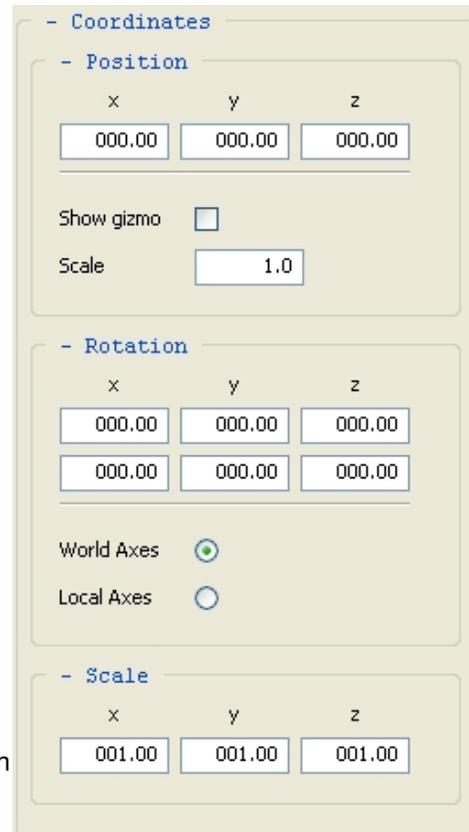
System and the *Local System*. Rotation about the axes of the *Local System* is important e.g. to rotate the object around some symmetry axis.

x, y, z

There are two rows of values for rotations. The top row indicates the rotation around the respective axes in the *World Coordinate System* while the bottom row shows the rotation in the *Local Coordinate System.* Which one of them is the active system is determined by the selection of the radio buttons below with the exclusive options of **World** or **Local Axes.**

Scale

You change the size of an object or sub-object by *Scaling* it. The scaling is performed in the local *Coordinate System*.

x, y, z

The values of the scaling of the objects or sub-objects that have been applied after they have been created or imported. The values indicate the scaling along each of the axes of the *Local Coordinate System.*

Node List

The *Node List* shows the names and some of the attributes of the *node.* The *Remove Button* allows a *node* to be deleted. Before finally deletion of a *node* a confirmation dialog is opened. Note that after confirmation this operation can not be undone.
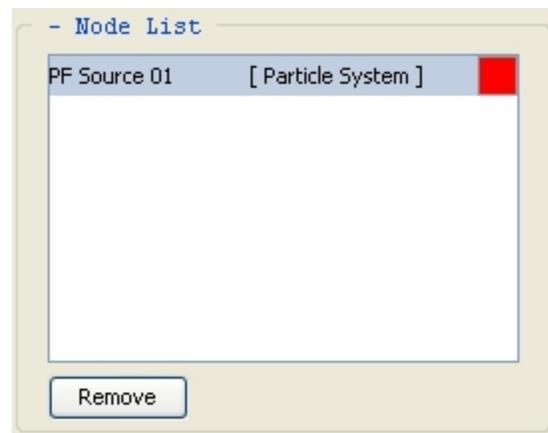
Particle Systems

In the *Particle System Panel* you manage the some properties of the *Particle System* that you have imported or created in the *3D Module.* One important function of the *Particle System Panel* is to set the velocities that the *Particles* will have. For *Particle Systems* that have been imported as such these velocities may already be included. However, with these may be overridden using the parametric *velocity field* applied here. The parametric velocities will be applied only if the *Calculate Velocities Checkbox* is set.

Properties

The *Properties Sub-Panel* shows the *Brightness, Flux, Color*, *Number of Particles* and the *Velocity Parameters.*

Brightness

The *Brightness* is a relative measure of the brightness of an individual *Particle.* Its function in the model is that of the *emissivity* in a gaseous nebula.

Flux

The *Flux* is the total emission coming from this *Particle System.* It is the product of the *Brightness* and the *Number of Particles* (*# Particles*)*.*

Color

The color of the *Particle System.*

 # Particles

The number of *Particles* in the *Particle System.*

Calculate Velocity

Check this tick box if you wish that the velocities of the *Particles* are to be calculated as a function of the *position vector* of the *Particle* in the *World Coordinate System.* If this tick box is unchecked the velocities are either zero (for *Particle System* created in the *3D Module*) or they are the velocities that have been imported with the external *Particle System.*

There are three different types of velocity field from which you can choose using the drop-down list: RADIAL EXPANSION, DISK ROTATION and SOLID BODY ROTATION.
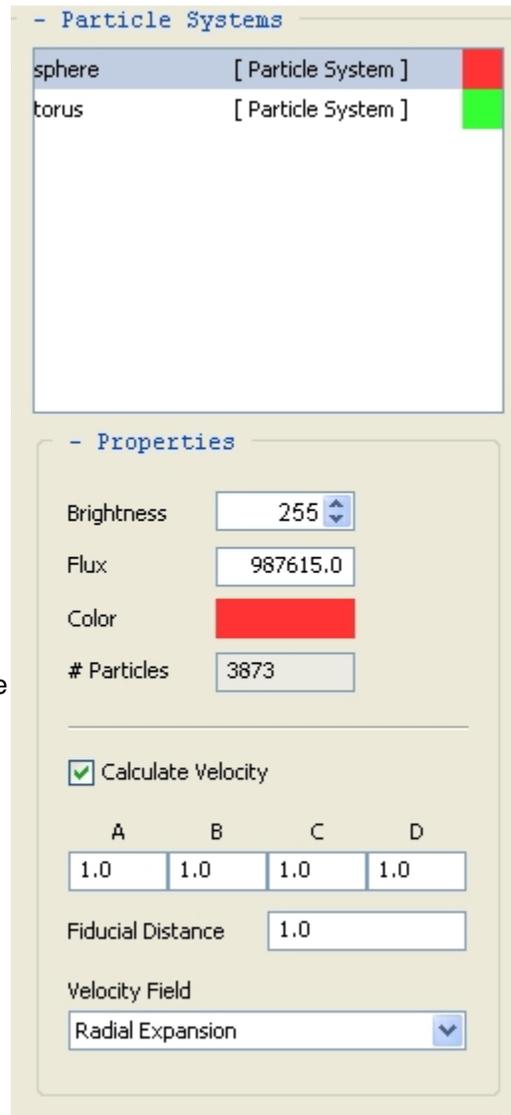
RADIAL EXPANSION

The *velocity vector* $\vec{v}$ is always directed along the *position vector* in the *local coordinate system* and obeys the following formula:

$$\vec{v} = \hat{p}\,[A + B\,p/r + C\,(p/r)^D]$$

where **A, B, C** and **D** are the coefficients. *p* is the particle distance from the origin of the *local coordinate system* (length of the position vector) and *r* is a reference distance.

Using suitable combinations of these coefficients, a large variety of velocity fields can be generated. Some particularly important examples are the following:

Constant radial velocity:

A = velocity

B = C = D = 0

"Hubble type" flow (i.e. velocity vector is proportional to the position vector):

A = C = D = 0

B = velocity at reference distance r

If you are unfamiliar with power-law functions, you might want to use an external plotting program to plot the velocity and adjust the coefficients before introducing them in *Shape*.

DISK ROTATION

The *velocity vector* $\vec{v}$ is always perpendicular to the *position vector* and the z-direction of the *local coordinate system*. It obeys the following formula:

$$\vec{v} = \hat{p}_{xy}[A + B\,p/r + C\,(p/r)^D]\times\hat{z}$$

where **A, B, C** and **D** are the coefficients. *p* is the particle distance from the origin of the *local coordinate system* (length of the position vector) and *r* is a reference distance . $\hat{p}_{xy}$ is the unit position vector of the particle in the xy-plane of the *local coordinate system. Note that the rotation axis of the rotator is restricted to the z-axis of the local coordinate system.*

Using suitable combinations of these coefficients, a large variety of velocity fields can be generated. Some particularly important examples are the following:

Keplerian disk:

A = B = 0

D = velocity at the reference distance r

D = - 0.5

If you are unfamiliar with power-law functions, you might want to use an external plotting program to plot the velocity and adjust the coefficients before introducing them in *Shape*.

SOLID BODY ROTATION

The *velocity vector* $\vec{V}$ is always perpendicular to the *position vector* and the z-direction of the *local coordinate system*.

For the solid body rotation only coefficient **A** is used and it corresponds to the velocity at the reference distance r. *Note that the rotation axis of the rotator is restricted to the z-axis of the local coordinate system.*

How to make *channel maps*

A *channel map* is an image which includes only those regions of a nebula which have a radial velocity (Doppler-shift) within a given range. If the range of velocities in a *channel map* is small compared to the whole range of velocities that are present in the nebula, these maps allow to "dissect" the nebula along the line of sight. However, similar to the case of *position-velocity diagrams* this is only a good view of the spatial dimension, if there is a reasonably regular relation between the spatial and the velocity domain. However, even if there is no such relation, the *channel map* still conveys important information on the structure and kinematics of the object.

To produce a *channel map* centered at a velocity of -5 and a range of 0.3 on both sides of the velocity center, follow these steps:

1. In the *Velocities Parameter Tab* set the *Center* to -5 and the *Range* to 0.3.

2. It is convenient to enable the *Rainbow CT* and set the *CT Range* to the same as the velocity *Range,* i.e. 0.3. This allows to visualize the *channel map* not only in the *Chan Map* image, but also in the *RB Shift* image. The *Rainbow CT* will mark the regions selected by the *channel map* in the grey-scale version of the full image. In some models this allows a clearer insight in the location of the selected region within the whole object.
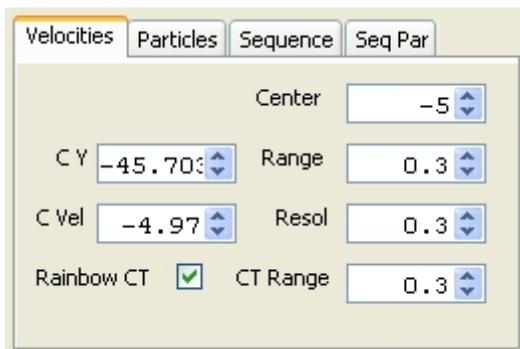
3. Choose any other parameter according to you needs.
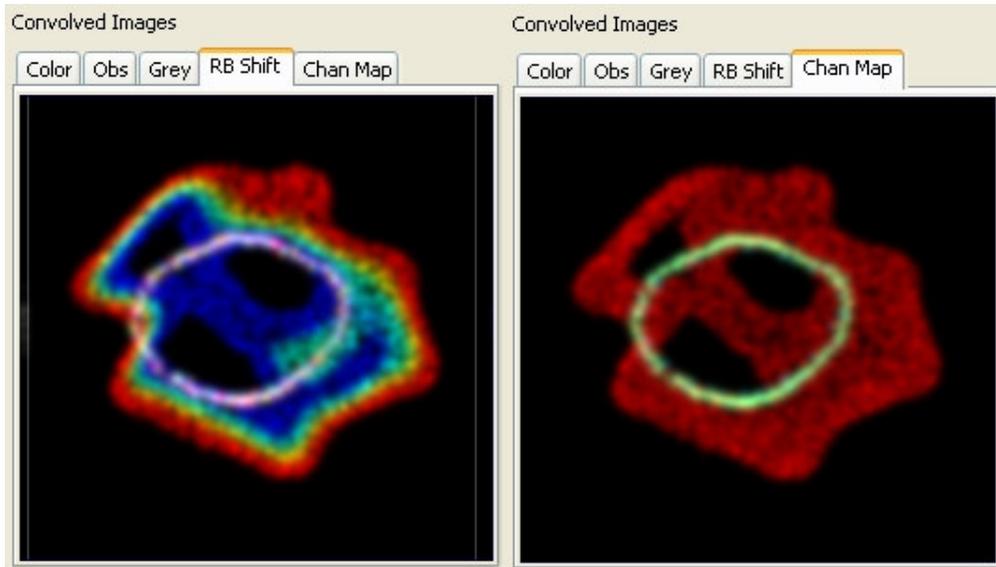
4. If you wish to produce a sequence of *channel maps* stepping through the full range of the object, do the following in addition to the previous steps:

- Select the *Enable Checkbox* in the *Sequence Parameter Tab.*

- Set the *Center* value to the initial velocity of the sequence (typically the same as the highest blue-shift that is present in the object, which can be read from the *pv-diagram* using the cursor).

- In the *Sequence Parameter Tab* set the *Start Frame* = 0

- Set the *Step Size* to the difference between the *Center* velocities of adjacent *velocity channels.*

- The *Num Steps* parameter should be the difference between the highest and lowest radial velocities divided by the *Step Size.*

- Set the *Image #* to 0.

- Type a suitable *Sequence Name* in the *text field* at the bottom of the *Sequence Parameter Tab.* The results of this sequence will be written into a sub-directory with the same name. From these image sequences, animations can be assembled using external movie editing programs. For a simple playback of the sequence, choose the *playback direction* and press any keyboard key or repeatedly click on the *playback button* in the *Sequence Parameter Tab*.

Particle system templates

In this tutorial you will learn how to use *Shape* starting with a particle system template that you download from the *ShapeSite*.

You go to the *Templates* area of the *ShapeSite* using your favorite Internet browser and download to your working directory the *particle system* template you wish to use. If compressed and packed, unpack the downloaded file. A sub-directory is created with several files, which at least contain the files "shape j3d.dat", "shape j3d.inp" and an info file with the extension ".txt", which contains some information on the object model and how to use the template. Some template packages may contain the original model files from the external 3D modeling software. This allows you to modify the template data as needed. Some changes can be done directly in the *3D Module* of *Shape.* The information in the info file may be as follows:

1. Download and unpack the zip-file in a directory of your choice

2. Start *Shape*

3. Start a new Project:

File > New Project

Insert a project name and path.

4. Import the particle system data:

File > Import > Data

With the Open Dialog navigate to the directory where you unpacked the template. Select the file shape j3d.dat. Verify that the data have been imported correctly. Open the 3D Interface using the 3D button on the tool bar. If the particle system is visible, the data have been imported properly.

5. Import the project parameters:

File > Import > Parameters

With the Open Dialog navigate to the directory where you unpacked the template. Select the file shape j3d.inp. The parameters on the Parameter Tabs should have changed. To verify, run *Shape* by clicking on the *Shape Button* on the *Tool Bar*. The rendering dialog should open and after finishing the render the image panels should show the rendered image and spectral information.

6. You can now manipulate the observing parameters for the object in the Parameters Tabs and/or manipulate the data using the *3D Module*.

7. If you wish to modify the original data, you may use *3DStudio Max* to open and modify the file bipolar sym 1.max. To export the result to shape j3d.dat use the export script Sh out j3d.ms.

External VRML and *.obj* 3D model files

The Virtual Reality Markup Language (*VRML*) format is a common scene description format. It can be used to import objects created in external 3D modeling software into *Shape* via the *3D Module*. Note that velocity information is not included in the VRML format at must be generated with the *3D Module* options that are available for this purpose. Let us assume that you already have a file with your object in the *VRML format* called "object.wrl" (wrl is the typical file extension for VRML files). To import the file go to the menu bar of the 3D Module. Choose

File > Import > VRML

Now a file selection dialog opens. Go to the directory in which your "object.wrl" file is located and select the file. Confirm the selection by clicking on the *Open* button of the dialog. Your object should now be visible in the *viewports* of the *3D Module* as a shaded surface. In order to be renderable in *Shape*, this surface object must be converted into a *particle system.* To do this select the *object node* in the *Node List* and then go to the menu bar of the 3D Module and choose

Groups > Create particle system

The shaded surface has now turned into a collection of points, one at each vertex of the original surface object. You may now change the *color* and *velocity structure* of the *particle system* in the corresponding panels on the right of the *3D Module*. You can also apply any of the *Transformations* available on the *Tool Bar.* Now you can render the object choosing suitable parameters in the MCI and by clicking on the *Shape Button.*

A similar procedure can be followed to import *.obj* files.

Particle systems from external 3D modeling software

Modeling with *Shape* is most detailed harnessing the powerful *particle systems* that come with some external 3D modeling suites. *ParticleFlow* in *3DStudio Max* or similar systems in other modeling suites can be controlled in detail either interactively or, most precisely, via script programming. A script for exporting *ParticleFlow* data in the *Shape* particle data format (see below) is provided in the *download area* of the *ShapeSite.* Check the corresponding download area for export scripts for other 3D modeling packages.

Once you have exported your particle data file, it can be imported into *Shape.* This is done via the *Menu Bar* in the *MCI.* From the *Menu Bar* choose

File > Import > Data

Now the file selection dialog opens. Select and confirm the import of the file. If the *3D Module* is open, the particle system should be visible in the viewports and the *Nodes Panel.* It can now be further processed (changing color, brightness, particle distribution, velocity field, etc.) in the *3D*

Panel or directly rendered with the *Shape Button* of the *MCI*. Before the data are rendered, they are written to the *shape j3d*.dat file, which is read by the *renderer.* If you save the *project*, the imported data are saved with the project.

Model templates

See the latest instructions on the *News Panel* at startup of *Shape.*

FILE FORMATS

The ASCII data formats of the particle system data and parameter files "shape j3d.dat" and "shape j3d.inp" are described. This allows users to write export scripts or programs for external modeling software. An example script for *3DStudio Max* which produces the file "shape j3d.dat" from selected particle systems of type *Particle Flow* can be downloaded from the *3DStudio Max* section of the link "External 3D modelers" on the *ShapeSite*. The file "shape j3d.inp" controls the parameters of the *Parameter Tabs* of the *MCI*. The file "shape j3d.inp" is written by the *MCI* when you render the object by clicking on the *Shape Button.* The data from both files are included in the *project file* when you save the project. Generally there is no need to externally modify the parameter file "shape j3d.inp". It's content in directly controlled via the *Parameters Tab.* However, note that the parameters are written to file only when you render or save the project.

***Note:* These file formats are subject to change with mayor new releases of *Shape.* Secondary releases will conserve these formats.**

General format of the file "shape j3d.dat":

- Main header
- First particle system header
- Particle data of first *particle system*
- Headers and particle data of any further *particle systems*

Details of file format (example):

Lines starting with "#" denote comments and are not read by the rendering engine. The two particle systems in the data file contain two particles each. The number of particle systems or particles per system is not restricted a priory. However, the rendering time is roughly proportional to the number of particles in the system.

```
# Multi-object particle positions for SHAPE
# Number of Systems:
2
# TIME:
0.
# Particle System ID:
PF Source 01
# Position of PS in World Coord:
0.0 0.0 0.0
# Color of PSystem (RGB Alpha)
0 255 153 255
# Number of Particles:
2
# N Part X Y Z VX VY VZ SIZE AGE AX AY AZ
1 1.10329 67.0187 -57.5515 0.158115 9.60459 -8.24782 1.0 5.0 0.0 0.0 0.0
2 -51.5906 18.4107 -42.7453 -7.39355 2.63847 -6.12592 1.0 5.0 0.0 0.0 0.0
# Particle System ID:
PF Source 02
# Position of PS in World Coord:
0.0 0.0 0.0
# Color of PSystem (RGB Alpha)
0 225 53 255
# Number of Particles:
2
# N Part X Y Z VX VY VZ SIZE AGE AX AY AZ
1 1.10329 67.0187 -57.5515 0.158115 9.60459 -8.24782 1.0 5.0 0.0 0.0 0.0
2 -51.5906 18.4107 -42.7453 -7.39355 2.63847 -6.12592 1.0 5.0 0.0 0.0 0.0
```

The detailed format of the "shape j3d.inp" file is as follows:

```
Project name
#Check boxes: Conv Grey, Conv Color, VelShift, Logscale, Verbose, Show Slit, Vel Slice, Opaque Star
true true false false false false false false
#ImCenterX ImRangeX
0.0 300.0
#ImCenterY ImRangeY
0.0 300.0
#VelCenter VelRange ColorTabRange
0.0 500.0 50.0 true
#Seeing VelResol
20.0 25.0
#gammaim gammaspec
1.5 2.5
#ImSizeSp.value
8
#SlitPosCenter SlitWidth SlitIntense
0.0 39.375 1.0
false false
#WorldRotX WorldRotY WorldRotZ
20.0 0.0 0.0
false false false
#LocalRotX LocalRotY LocalRotZ
0.0 0.0 0.0
false false false
#ParticleRadius PSType
20.0 1.0
#Do Sequence? Step Num Steps
false 0 1 0.0
```

Known issues and bugs

- If using a multi-monitor system, currently you can not change the viewpoint of a view panel while the 3D Module is displayed on a secondary monitor. The work around is to move the 3D Module to the main monitor and do the change there. Now you can move the 3D Module back and the new view will work.

- The *Shape View Panel* does not *Zoom Extent* on selecting the corresponding option from the right-click pop-up list.

Some external 3D Modeling Systems

This is only a small selection of commercial and free 3D modeling system around. Please feel free to find the one you like most. However, please note that in order to work with *Shape* the system needs at least capability to export to VRML format. It is highly desirable to have a good *Particle System* tool in combination with a scripting language that allows you to export particle positions and velocities in the format that *Shape* can read.

3DStudio Max - Data export script (*MaxScript*) for *Shape* is available at the *ShapeSite. 3D Studio Max* is a commercial 3D modeling suite. The *Particle Flow* system has been used extensively for modeling with *Shape.* Surface structures can be exported in VRML format, which can be imported into *Shape* and then converted into a *Particle System.*

Blender - Free 3D modeling suite that is included in some *Linux* distributions. Scripting language is *Python.* Currently there is no *Particle System* export script available (please write one and share it with the *Shape* user community).

Maya - Commercial 3D modeling suite. Free educational version is available with limited capabilities. However, these are not very restrictive and very well suited for usage with *Shape.* There is currently no export script (*MelScript*) available for *Particle Systems* (please write one and share it with the *Shape* user community).

Bibliography

Steffen, W., López, J.A., 2006, *Morpho-kinematic modeling of gaseous nebulae with SHAPE,* RevMexAA 42, 99-105

CREDITS:

Principal Investigator: Wolfgang Steffen

(Instituto de Astronomía, Universidad Nacional Autónoma de México (UNAM), Ensenada, BC, México)

Programming:

Wolfgang Steffen

Nicholas Koning (University of Calgary, Calgary, Canada)

User Manual: Wolfgang Steffen

Financial Support:

Universidad Nacional Autónoma de México, DGAPA, PAPIIT

Grants: IN108506, IN111803, IN112103

Consejo Nacional de Ciencia y Tecnología (CONACYT), México

Grants: 49447, 33660-E

University of Calgary, Calgary, Canada

Grant: Natural Sciences and Engineering Council of Canada awarded to Sun Kwok

Other scientific contributors:

José Alberto López García

(Instituto de Astronomía, Universidad Nacional Autónoma de México (UNAM), Ensenada, BC, México)