ImageStream

Version 6.00

User's Guide

by Dr Roger Nokes

February 2007

ImageStream

Version 6.00

User's Guide

February 2007

Department of Civil Engineering
University of Canterbury
Christchurch, NZ
roger.nokes@canterbury.ac.nz

Design and layout: Melody Callahan

1 Introduction

ImageStream is a simple image processing package aimed at experimental fluid mechanics applications. It has been developed in the Department of Civil Engineering at the University of Canterbury and has undergone continual development over a number of years.

ImageStream is based around the concept of an image sequence, which is, essentially, a video record of an experiment stored as a sequence of individual image files.

Manipulation of the images in an image sequence is allocated to transformation algorithms, known as filters. A filter takes the pixel intensities of a digital image as input, transforms these intensities according to its specific rules, and produces new pixel intensities as its output. A pipeline of filters can be defined whereby the output of each filter in the pipeline is fed into the next filter. Thus multiple transformations can be applied to the pixel data.

ImageStream has been designed to work with a range of digital image data types. Standard digital video produces images containing 32 bit RGB colour. More specialised cameras may generate 10 or 12 bit gray-scale images instead. And the image processing undertaken by *ImageStream* itself may produce real value data at each pixel. All of these data types are catered for, and standard image storage formats, JPEG, TIF, PNG, BMP etc, are used to store all data types on disk. The resulting images may or may not appear meaningful when displayed in standard image viewing software.

While the transformed images may be an end in themselves, it is more common for the user to want to inspect the pixel information in these transformed images. This is achieved by creating an intensity field based on a rectangular grid and providing various tools for manipulating and viewing this intensity field.

All of these features are described in detail in the two *ImageStream* documents. The first, the ***ImageStream* – System Theory and Design** Manual, provides a general introduction to image processing, particularly as it applies to experimental fluid mechanics, and describes in detail the theory underpinning the *ImageStream* implementation. The second is this manual, the ***ImageStream* User's Guide**. Here details of the user interaction mechanisms are provided.

ImageStream is designed according to the object model of software development. Both the underlying code and the user interface are consistent with this model. While the code is of little concern to the user, the structure of the interface is. The *ImageStream* system comprises a number of objects. The top level object is the application itself, while other types of object include image sequences, regions, filters, intensity fields and so on. The user interacts with these objects in a consistent way. Each object processes zero or more views of itself. These views display different aspects of the information stored within the object. For example, the image sequence object has a total of three views. The image view allows the user to view the images as if they were a video recording but with a number of useful enhancements. The filter view enables the user to create filters that can be used to transformation the pixel information in the sequence's images. And the details view provided basic information about the image sequence. For every object type one of the views acts as the default.

Objects within the system are generally displayed to the user in a list box. For example the *ImageStream* object keeps track of the image sequences currently opened within the system and displays these in one of its views, the image sequence list view. The user can interact directly with objects in a list box by double clicking the object in order to open the default view, or by right clicking on the object and selecting an action from the resulting popup menu.

Object creation is often accomplished through special dialog boxes that prompt the user for key information. Time-consuming actions, such as filtering an image sequence, are generally run in a separate thread and tracked with special monitor dialog boxes that inform the user of progress and allow the user to abort the operation if desired.

All of the usual graphical user interface components are present.

This manual

- provides the user with detailed information about the various objects present in *ImageStream*,
- explains the available object views, the information they display and the menu options each provides,
- introduces the popup menus and the commands they provide, and
- explains the dialog boxes and the parameters they require.

2 ImageStream

2.1 Description

An *ImageStream* object is the top-level object that is created when the *ImageStream* application runs. This object provides the user with the top level of user interaction, including the main menubar (see application view below).

The image sequences, intensity fields and other objects created by the application are managed by the *ImageStream* object.

2.2 Views

The *ImageStream* object provides four views. These are:

- application view
- image sequence list view
- intensity field list view
- field list view

2.2.1 Application view

Description

The application view is the parent window for the application. It includes the main menubar. When this view is closed the application exits.

Access

The application view is opened when the application starts.

Menus

Application Menu

- *Change look and feel* – this option provides the user with a choice of interface "looks and feels" or "skins". A look and feel defines how the various windows, controls etc are displayed to the user. The default look and feel will be that of the operating system under which the application is running. On a Windows machine the user

interface will look like any other Windows application. On a Macintosh the interface will conform to Apple standards (although the single menu bar characteristic of Macintosh applications is absent – each window having its own menu bar). The user will generally have the option of selecting two other looks and feels. One is called Metal and is a generic Java interface design. The second is Motif, which is the standard look and feel for some X Windows based UNIX systems. The user is free to switch between these different skins at any time. It is worth noting that the ImageStream application view will always retain the native title bar look and feel no matter what skin has been selected, and this is also true of most dialog boxes.

- *Run garbage collector* – attempts to retrieve unused memory in the system
- *Properties…* – allows the user to access the application properties. These include default directories and memory management parameters (see the **System Theory and Design** Manual)
- *Quit* – allows the user to exit the application. The user will be prompted to save unsaved data before exiting.

View Menu

- *Image sequence list view* – opens another image sequence list view. This view displays all of the image sequences (see chapter 3) currently opened in *ImageStream*.
- *Intensity field list view* – opens another intensity field list view. This view displays all of the intensity field objects (see chapter 7) currently opened in *ImageStream*.
- *Field list view* – opens another field list view. This view displays all of the field objects (see chapter 10) currently open in *ImageStream*.
- *Change background colour…* – allows the user to select a different background colour for the application window.

Sequence Menu

- *Open image sequence file…* – allows the user to open an image sequence object from a file with extension *.isq6*. The loaded object will be listed in the image sequence list view.
- *Load sequence from raw text file…* - allows the user to open an image sequence from a text file created under versions 4, 5 or 6 of the software. These files have extensions *.stx4, .stx5 and .stx6* respectively.
- *Create image sequence…* – allows the user to create a new image sequence.
- *Create batch process…* – allows the user to create a number of different batch processes. A batch process is essentially an

automated process whereby analyses that would normally be performed manually on an image sequence can be processed overnight (or at any other time) without user intervention. The first batch process allows the user to select a number of image sequence files and to have each of these sequences apply their filters to their images. The user selects the image sequence files, and the corresponding file names for the filtered images. The remaining two batch processes calculate an average colour or average real value image for each of the image sequence files selected. Again the user can choose the file in which the average image will be stored, once it is calculated. More details on these processes can be found in chapter 3.

Intensity Menu

- *Open intensity field file…* – allows the user to open an intensity field object from a disk file with extension *.int6*. The opened object will be listed in the intensity field list view.
- *Load intensity field from raw text file…* – allows the user to open an intensity field object from a text file that has been created under versions 3, 4, 5 or 6 of the software. These files have extensions *.itx3*, *.itx4*,.*itx5* and *.itx6* respectively.
- *Create concatenated intensity field …* – allows the user to create a new intensity field by concatenating a number of previously stored intensity fields. Details of concatenated intensity fields can be found in chapter 9.

Field Menu

- *Open field file…* – allows the user to open a field object (see chapter 10) from a disk file. The opened object will be listed in the field list view.

Image Menu

- *Display image…* – allows the user to view any image file. A dialog box containing the image is displayed. Text boxes record the position and pixel intensity as the cursor moves over the image. As *ImageStream* does not know what type of data is stored in the image, the 32bit value at each pixel is translated into an RGB value, a 10 bit gray-scale value, a 12-bit gray-scale value, and a real value.
- *Create constant colour image…* – this option allows the user to create an image file where each pixel in the image has a constant intensity. The user can specify the type of data that is stored in the image (RGB, gray-scale etc) and the intensity value.

Info Menu

- *Memory…* – the user can have a real time display of the system memory by selecting this menu item. The total memory available, and the total memory currently unused, are updated every second while the dialog box is displayed. The user can run the garbage collector from this dialog box.
- *About…* – provides information on the software designer and the software version.

2.2.2 Image sequence list view

Description

The image sequence list view provides a list of the image sequences (see chapter 3) currently opened in *ImageStream*. The name of each image sequence, together with the name of the file in which it is stored, is displayed. The user can interact directly with these objects by double clicking them to open their default view, or right clicking to display their popup menu.

Access

The image sequence list view is opened through the *ImageStream View* menu. It is automatically opened on start-up and is positioned at the top left of the application window.

2.2.3 Intensity field list view

Description

The intensity field list view provides a list of the intensity fields (see chapter 7) currently opened in *ImageStream*. The name of each intensity field, together with the name of the file in which it is stored, is displayed. The user can interact with these objects directly by double clicking to display the default view for the field, or right clicking to display the object's popup menu.

Access

The time series list view is opened through the *ImageStream View* menu. It is automatically opened on start-up and is positioned at the middle left of the application window.

2.2.4 Field list view

Description

The field list view provides a list of the fields (see chapter 10) currently opened in *ImageStream*. The user can interact directly with these objects by double clicking them to open their default view, or right clicking to display their popup menu.

Access

The field list view is opened through the *ImageStream View* menu. It is automatically opened on start-up and is located at the bottom left of the application window.

3 Image Sequences

3.1 Description

Image sequences are the starting point for image processing in the *ImageStream* system. An image sequence represents a complete video record comprising a sequence of still images captured at a fixed frame rate. The user provides scales that enable the pixels in an image to be converted into physical coordinates in mm.

The images in an image sequence may be in a number of image formats. These formats are BMP, JPG, TIF, PNG and GIF. At the same time the data stored within these image files can be of a variety of types. If the images have been captured by a standard digital video camera then the 32 bit colour information stored at each pixel comprises three 8 bit values corresponding to red, green and blue intensities, and an additional 8 bit value corresponds to the alpha setting, which is not used in *ImageStream*. On the other hand, images that have been captured by non-standard scientific digital video cameras may represent 10-bit or 12-bit gray-scale images. As far as the author is aware there is no standard format for this intensity information so *ImageStream* uses its own standard, where each pixel is assumed to store a 32 bit RGB value. The most significant 8 bits in the gray-scale signal are stored in the red intensity, while the remaining bits are stored in the green intensity. The blue and alpha values are ignored in this case. Finally, *ImageStream* itself may produce images that contain floating point information at each pixel (for example when light intensity is converted to dye concentration). Such images are referred to as storing real values. In this case the IEEE 32 bit floating point representation of the real value is copied directly into the 32 bit colour intensity. When an image sequence is created the type of data stored in the images must be specified.

All images in an image sequence must have the same dimensions (measured in pixels). This requirement is checked when the image sequence is created, and any images that do not conform are excluded.

Each image sequence is allocated a name by the user and a textual description can also be provided.

3.2 Views

The image sequence provides three different views of the data it stores. These are:

- details view
- filter view
- image view

3.2.1 Details View

Description

The details view of an image sequence displays the key details about the sequence, including the number of frames, image size (in pixels), scales, time step, description, image type etc. A list box displays the list of images in the clip and allows access to their popup menus and views. If the image sequence has been created through a transformation (see under popup menu) then the transformation is listed in a separate list box.

The details view allows the user to change a number of the image sequence parameters. This includes the name of the sequence, the description, the x and y scales for converting from pixels to mm, the time step between frames, and the type of data stored in the images. The user must press the *Save changes* button for these changes to take effect.

Access

This view is accessed through the popup menu of an image sequence. Image sequences are displayed in the *ImageStream* image sequence list view.

3.2.2 Filter view

Description

The filter view gives the user control over the filters defined for an image sequence.

The filters defined for an image sequence can be viewed as a pipeline through which pixel data from each image can be passed

and processed. The pixel intensities from each image are passed to the first filter which processes these intensities and produces a transformed image. The output pixel data from this filter is passed to the next filter in the pipeline and so on until all filters have been applied. The net effect of the application of these filters is a final image. This pipeline must be consistent in order for it to be able to work correctly. In a consistent pipeline the data type of the image sequence must match the input type of the first filter in the pipeline, and then the output type of each filter must match the input type of the next filter.

The filters currently defined are displayed in the list box at the top of the view. Alongside the filter name the input and output types of the filter are displayed. If there are any inconsistent filters within the pipeline (ie the input type of that filter does not match the output type of the preceding filter) their names and input and output types are displayed in red instead of black. Such inconsistencies must be corrected before the filter pipeline may be applied.

The buttons at the bottom of the view provide the following functions:

- *New* – creates a new filter object and adds it to the pipeline. A dialog box is displayed that prompts the user for the choice of filter (see chapter 6).
- *Open* – the user can load a previously saved filter from a disk file with extension *.flt* and add it to the pipeline.
- *Insert before*…– allows the user to load a previously saved filter from a disk file with extension *.flt* and insert it before the currently selected filter in the pipeline.
- *Delete* – deletes the currently selected filter in the pipeline.
- *Clear* – this button clears all filters from the pipeline. The user is prompted to confirm this action.
- *Apply filters* – all filters in the pipeline are applied to the image sequence. Each image is filtered through the pipeline and stored in a numbered image file whose name and format (BMP, PNG etc) is specified by the user. The result is a new image sequence that appears in the image sequence list with a name specified by the user.

Access

This view is accessed through the popup menu of an image sequence. Image sequences are displayed in the *ImageStream* image sequence list view.

3.2.3 Image view

Description

The image view provides the user with an environment in which to inspect the raw or filtered images in an image sequence. If these images store real data then the value of this view is limited.

The view displays an image from the sequence beneath a number of toolbars. The toolbars provide the user with a simple video replay facility whereby the images can be played forward or backward, one frame at a time or continuously. It should be noted that the continuous motion replay is unlikely to occur at real speeds due to the time taken for each image to be loaded from disk. However, the user can select a time delay between frames, which can be used to slow down the replay if they wish.

Two of the toolbars provide the user with information about the cursor location and the intensity of the pixel over which the cursor resides. The view interprets the intensity at each pixel according to the data type specified when the sequence was created.

Details of the toolbars are given next.

Video toolbar

The top toolbar is the video toolbar. This toolbar provides the video controls that allow the user to control the video features of the view.

Controls	Description
Frame displayed	This combo box displays the frame currently displayed in the view. The frame number and time are both shown.
 *Go to next image*	Steps the view forward one frame at a time.
 *Go to previous image*	Steps the view backward one frame at a time.
 *Play sequence*	Plays the sequence of images forwards in time until the last frame is reached. The speed of animation is determined by the setting in the animation time delay field
 *Replay sequence*	Plays the sequence of images backwards in time until the first frame is reached. The speed of animation is determined by the setting in the animation time delay field.
 *Pause*	Pauses the animation and is only available if one of the play or replay buttons is currently pressed
 *Go to end of sequence*	Moves to the end of the sequence, displaying just the final frame.
 *Go to beginning of sequence*	Moves to the beginning of the clip displaying the first frame.
Animation time delay (ms)	This text field allows the user to specify the time delay between frames during an animation. This figure is in milliseconds and by default is set to the time step of the image sequence. Increasing this figure allows a slow motion relay to be observed.

Cursor toolbar

The cursor toolbar, second from the bottom, displays the location of the cursor on screen and indicates the current drawing tool.

- *x* – This text box displays the x coordinate (in mm) of the current cursor position. The box is updated as the cursor is moved over the image. It cannot be edited by the user.
- *y* – This text box displays the y coordinate (in mm) of the current cursor position. The box is updated as the cursor is moved over the image. It cannot be edited by the user.
- *Drawing tool* – This text box displays the name of the currently selected drawing tool. This tool is used for defining regions in the view (see below and chapter 5). The tool can be changed using the *Region* menu.

Pixel toolbar

The pixel toolbar sits at the bottom of the toolbar area. This toolbar displays the pixel intensities corresponding to the current cursor location.

- *Red* – This text box displays the red intensity of the pixel currently under the cursor if the image stores RGB data. It displays the word "undefined" if the data type is not RGB.
- *Green* – This text box displays the green intensity of the pixel currently under the cursor if the image stores RGB data. It displays the word "undefined" if the data type is not RGB.
- *Blue* – This text box displays the blue intensity of the pixel currently under the cursor if the image stores RGB data. It displays the word "undefined" if the data type is not RGB.
- *Gray (10 bit)* – This text box displays the 10 bit gray scale intensity of the pixel currently under the cursor if the image stores 10 bit gray scale data. It displays the word "undefined" if the data type is not 10 bit gray scale.
- *Gray (12 bit)* – This text box displays the 12 bit gray scale intensity of the pixel currently under the cursor if the image stores 12 bit gray scale data. It displays the word "undefined" if the data type is not 12 bit gray scale.
- *Real* – This text box displays the real value stored at the pixel currently under the cursor if the image stores floating point data. It displays the word "undefined" if the data type is not real.

Creating Regions

Regions (see chapter 5) are a tool for selecting areas within an image. Regions are created within the image sequence image view by using a range of drawing tools selectable within the *Region* menu of the view. Four different region geometries are available – rectangle, circle, polygon and freehand drawing (in essence a high order polygon). For each geometry the region inside or outside the figure can be defined as the region of interest. The default drawing tool is the Internal Rectangle. Thus the region is defined as the area within a rectangle drawn on screen. If the drawing tool was the External Rectangle then the region would be defined to be the area outside the rectangle. The colour of the region's outline distinguishes between the internal and external setting. Internal regions are drawn in dark blue, and external regions are drawn in red.

Details regarding how the different drawing tools are used is provided in chapter 5, together with information on the manipulation of regions already created. It is important to note that the image display area must have the mouse and keyboard focus during region manipulation. A small rectangular indicator in the bottom right hand corner of the view is green when the area has the focus and red when it does not. To regain the focus simply click the mouse in the image display area.

Regions have a number of uses. The first is in the definition of certain filters that are based on regions. The second is in the creation of intensity fields. The regions allow certain areas within an image to be excluded from the intensity field calculation (see chapter 8).

The image view is the default view for an image sequence.

Access

This view is accessed through

- the popup menu of an image sequence. Image sequences are displayed in the *ImageStream* image sequence list view.
- double clicking on an image sequence.

Menus

View Menu

- *Show raw image…* – when selected the raw images of the image sequence are display in the view.
- *Show filtered image…*– when selected the images in the sequence are passed through the filter pipeline and the images resulting from these transformations are displayed. If the filter pipeline is inconsistent the raw images are displayed.

Region Menu

- *Display regions…* – displays all of the currently defined regions in a list box.
- *Load regions…*– previously stored regions are loaded from a disk file and added to the currently defined regions. Regions are stored in a file with a *.rgn* extension.
- *Save regions as…*– saves all of the currently defined regions in a *.rgn* disk file.
- *Delete all regions* – deletes all currently defined regions. The user is prompted to confirm this option.
- *Delete selected regions* – deletes all selected regions.
- *Select all regions* – selects all regions.
- *Unselect all regions* – unselects all regions.
- *Select <-> unselect* – unselects all selected regions and vice versa.
- *Invert selected regions* – converts all selected regions that are interior regions into exterior regions and vice versa.
- *Invert all regions* – converts all regions that are interior regions into exterior regions and vice versa.
- *Draw internal rectangle* – sets the current drawing tool to an internal rectangle. This is the default.
- *Draw internal circle* – sets the current drawing tool to an internal circle.
- *Draw internal polygon* – sets the current drawing tool to an internal polygon.
- *Draw internal freehand* – sets the current drawing tool to an internal freehand.
- *Draw external rectangle* – sets the current drawing tool to an external rectangle.
- *Draw external circle* – sets the current drawing tool to an external circle.
- *Draw external polygon* – sets the current drawing tool to an external polygon.
- *Draw external freehand* – sets the current drawing tool to an external freehand.

3.3 Popup menu

The image sequence provides the following options in its popup menu.

- *Save as…* – saves the image sequence in a new file. All image sequence files have the extension *.isq6*. The new file name is displayed next to the image sequence name in the *ImageStream* image sequence list view.
- *Save* – saves the image sequence to disk. If the image sequence currently has no associated file then the user is prompted for a file name. All image sequence files have the extension *.isq6*.
- *Save as raw text…* – saves the image sequence in a text format file that is readable by later versions of *ImageStream*. The text file has a *.stx6* extension.
- *Change name…* – allows the user to change the name of the image sequence. This name change will be reflected in the *ImageStream* image sequence list view. This action can also be performed through the details view.
- *Edit description…* – allows the user to change the description of the image sequence. This action can also be performed through the details view.
- *Delete frames…* – this option allows the user to delete frames (images) from an image sequence. The first and last frames to be deleted are selectable by the user. The frames are renumbered after the deletion if appropriate. This action is drastic and the user is prompted for confirmation.
- *Open details view* – opens the details view.
- *Open filter view* – opens the filter view.
- *Open image view* – opens the image view. This is the default view.
- *Apply filters…* – applies the filter pipeline to the images in the sequence. The user is prompted for the name of the filtered image sequence, and a filtered image filename. A 5 digit number is appended to the end of this filename to differentiate between the filtered images, and the extension of this file specifies the format in which the images are to be stored.
- *Create intensity field…* – creates an intensity field from the images in the sequence. See chapters 7 and 8.
- *Create transformed image sequence…* – this option allows the user to create a transformed image sequence. See chapter 4.
- *Close all views* – closes all open views for this object.

- *Close* – removes the image sequence from *ImageStream*. All views are closed. If the image sequence has unsaved data then the user is prompted to save this data before closure.

3.4 Creating image sequences

Image sequences are created by selecting *Create image sequence* from the *ImageStream Sequence* menu. A dialog box is displayed that prompts the user for the following key information.

3.4.1 Sequence data

The user must supply the following image sequence data.

- *Image sequence title* – the sequence name, used to identify the sequence from all others in the *ImageStream* image sequence list view.
- *Image type* – the type of data stored in the images. This can be RGB colour, gray scale (10bit), gray scale (12 bit), or real values.
- *Time step(s)* – the time interval, in seconds, between images in the sequence. This must be the same for all frames.
- *x scale* – the factor used to convert from pixel space to physical space in the x direction. The units are mm/pixel.
- *y scale* – the factor used to convert from pixel space to physical space in the y direction. The units are mm/pixel.
- *Width* – this field cannot be edited by the user and displays a value of –1 until the image files have been specified. In this case the width (in pixels) of the first image is displayed here.
- *Height* – this field cannot be edited by the user and displays a value of –1 until the image files have been specified. In this case the height (in pixels) of the first image is displayed here.

3.4.2 Image file selection

In this tab the user chooses the files from which the sequence is to be created. Each file corresponds to a frame in the sequence, and they must be listed in the same order as the frames will appear in the sequence. The files must be JPEG (.JPG), GIF, PNG, BMP or TIF. The selected files are displayed in a scrollable list box.

Management of the files is controlled through the four buttons below the list box. They perform the following tasks.

- *Add* – displays a file dialog box from which the user can select one or more files. Typically the user will select a complete sequence of GIF, JPEG, PNG, BMP or TIF files from the computer's file system.
- *Insert before* – allows the user to insert one or more files before a file selected in the list box. Again a file dialog box is displayed from which the user can select the files to be inserted.
- *Delete* – deletes all selected files in the list box. Multiple files can be selected using SHIFT+click.
- *Edit list* – allows the user to select a subset of the files to be used. This subset is defined as every n^{th} file in the list starting at a particular file. This is particularly useful when the flow is such that the video capture rate is unnecessarily fast. Thus one frame in every ten could be selected for example.

The user can display one of the images before creating the image sequence by double clicking on the file name.

Once the user presses the OK button to create the image sequence the file of selected files is checked for dimensional consistency with the first image in the list. A monitor dialog box is displayed showing the progress of this integrity check. Any images that do not conform to the dimensions of the first image are excluded from the image sequence and the monitor displays that a bad image has been identified and sounds a beep to alert the user.

4 Image Sequence Transformations

4.1 Description

The processing of images in an image sequence is generally performed by the filters defined for that image sequence. However, there are a number of transformations of the image sequence data that does not conform to the concept of a filter. These transformations are defined as image sequence transformations and are accessed through the popup menu of the image sequence.

The result of applying a transformation to an image sequence is to produce a new image sequence. This new sequence may contain only one image, or it may contain the same number of images as the original sequence. The images contained in the new sequence may have the same dimensions as the original images or they may not. It is clear that, these types of transformation do not fit into the standard definition of a filter which takes each image in an image sequence and converts it into a new image of the same size.

To perform a transformation on an image sequence select *Create transformed image sequence…* for the image sequence popup menu. This option will display a dialog box in which the user can choose a name for the new sequence, and select the desired transformation. For each transformation a Parameters button allows the user to specify the transformations parameters, as described in the following sections.

Currently six image sequence transformations are defined. These are described below.

4.2 Convert format transformation

The convert format transformation simply takes the images in the original sequence and converts them to a user specified format. Thus, for example, images originally in a BMP format could be compressed into a JPG format. The parameters for this transformation are:

- *Filename* – the user can select a filename for the converted images by clicking on the Browse button. A standard convention is used

here. For each image in the original sequence a 5 digit number is appended to the specified filename (minus the extension) so that the converted images are stored in a number of sequential files with the same name but different numbers. The extension of the filename selected by the user specifies the desired format.

4.3 Trim transformation

The trim transformation allows the user to trim the images in the sequence, thus producing smaller, rectangular sub-images. This is particularly useful for saving storage space, and processing time if only a portion of the captured images is of interest. The parameters for this transformation are:

- *Filename* – the user can select a filename for the converted images by clicking on the Browse button. The standard convention described in section 4.2 is used here.
- *Left inset (pixs)* – this parameter specifies the number of pixels on the left hand side of the image that will be trimmed. A value of 0 ensures no pixels are trimmed.
- *Right inset (pixs)* – this parameter specifies the number of pixels on the right hand side of the image that will be trimmed. A value of 0 ensures no pixels are trimmed.
- *Top inset (pixs)* – this parameter specifies the number of pixels at the top of the image that will be trimmed. A value of 0 ensures no pixels are trimmed.
- *Bottom inset (pixs)* – this parameter specifies the number of pixels at the bottom of the image that will be trimmed. A value of 0 ensures no pixels are trimmed.

4.4 Average colour transformation

The average colour transformation is applicable to any image sequence containing RGB images. The transformation produces a single image which contains the average colour of the original image sequence each pixel. Thus the red intensity is the average of all of the red intensities of the original sequence and so on.

The user can choose to apply the filter pipeline before performing this transformation but this is only possible if the pipeline produces an RGB image. The parameters for this transformation are:

- *Average filename* – the user can select a filename for the average colour image by clicking on the Browse button.

- *Apply filters* – when this checkbox is selected the filter pipeline is applied before the average image is produced. When not selected the raw images are processed.

4.5 Average colour constant velocity transformation

The average colour constant velocity transformation is similar to the average colour transformation in that a single average colour image is created from an image sequence containing RGB images. However, before this averaging is done the images in the sequence are translated in space according to a user specified velocity. This velocity is meant to simulate a constant movement of the image, and the transformation is an attempt to remove that movement from the images. Thus a positive velocity in the x direction causes the images to be moved to the left, while a positive velocity in the y direction causes the images to be moved downwards and so on. The result of this transformation is to create a larger image than those in the original image sequence, and care must be taken to ensure that the resulting image isn't unwieldy.

The specified velocity is unlikely to cause a pixel in the original image to lie exactly on a pixel in the transformed image. The user can choose whether to "snap" the transformed pixel onto the new image (in other words place it at the nearest pixel location) or to use a weighted average of the pixel intensities of the pixels that partially overlap each pixel in the new image.

The user can choose to apply the filter pipeline before performing this transformation but this is only possible if the pipeline produces an RGB image. The parameters for this transformation are:

- *Average filename* – the user can select a filename for the average colour image by clicking on the Browse button.
- *x velocity(mm/s)* – the horizontal velocity used in the transformation.
- *y velocity (mm/s)* – the vertical velocity used in the transformation.
- *Subpixel accuracy* – when this check box is selected a weighted average of intensities of pixels overlapping each pixel in the transformed image is used to calculate the new pixel intensity.
- *Apply filters* – when this checkbox is selected the filter pipeline is applied before the average image is produced. When not selected the raw images are processed.

4.6 Average value transformation

The average value transformation is very similar to the average colour transformation. However, here the original image sequence must store real value images and the transformation produces a single image which contains the average value of the original images at each pixel.

The user can choose to apply the filter pipeline before performing this transformation but this is only possible if the pipeline produces an real value image. The parameters for this transformation are:

- *Average filename* – the user can select a filename for the average value image by clicking on the Browse button.
- *Apply filters* – when this checkbox is selected the filter pipeline is applied before the average image is produced. When not selected the raw images are processed.

4.7 Average value constant velocity transformation

The average value constant velocity transformation is similar to the average value transformation in that a single average value image is created from an image sequence containing real value images. However, before this averaging is done the images in the sequence are translated in space according to a user specified velocity. This velocity is meant to simulate a constant movement of the image, and the transformation is an attempt to remove that movement from the images. Thus a positive velocity in the x direction causes the images to be moved to the left, while a positive velocity in the y direction causes the images to be moved downwards and so on. The result of this transformation is to create a larger image than those in the original image sequence, and care must be taken to ensure that the resulting image isn't unwieldy.

The specified velocity is unlikely to cause a pixel in the original image to lie exactly on a pixel in the transformed image. The user can choose whether to "snap" the transformed pixel onto the new image (in other words place it at the nearest pixel location) or to use a weighted average of the pixel intensities of the pixels that partially overlap each pixel in the new image.

The user can choose to apply the filter pipeline before performing this transformation but this is only possible if the pipeline produces an RGB image. The parameters for this transformation are:

- *Average filename* – the user can select a filename for the average value image by clicking on the Browse button.
- *x velocity(mm/s)* – the horizontal velocity used in the transformation.
- *y velocity (mm/s)* – the vertical velocity used in the transformation.
- *Subpixel accuracy* – when this check box is selected a weighted average of intensities of pixels overlapping each pixel in the transformed image is used to calculate the new pixel intensity.
- *Apply filters* – when this checkbox is selected the filter pipeline is applied before the average image is produced. When not selected the raw images are processed.

5 Regions

5.1 Description

Regions are simply geometric shapes defined in two-dimensional space that can be used in a variety of settings within Image*Stream*. Regions can be created within a number of views, for example the image sequence image view or some of the field views described in chapter 10. Six different region shapes can be created – points, lines, rectangles, circles, polygons and freehand figures (that are essentially many-sided polygons). Regions, other than the point and line regions, have an additional property. They either define the space internal to the shape, or the space external to it. Thus an internal rectangle defines the space within the four sides of the rectangle, while an external rectangle includes all space excluding that which lies within its four sides. The colour of the region's outline distinguishes between the internal and external setting. Internal regions are drawn in blue, and external regions are drawn in red.

Regions are drawn and manipulated in the same way, independent of the view in which they appear.

Regions are drawn in the following way.

- *Point* – the mouse is moved to the location of the point and the SHIFT key is held down while the left mouse button is clicked.
- *Line* – the left mouse button and SHIFT key are pressed when the cursor is located at one end of the line. The mouse is dragged (ie left button pressed) to the other end of the line and the button released. A rubber band is drawn during the dragging action. The SHIFT key need not be held during the drag process.
- *Rectangle* – the left mouse button and SHIFT key are pressed when the cursor is located at one corner of the rectangle. The mouse is dragged (ie left button pressed) to the opposite corner of the rectangle and the button released. A rubber band is drawn during the dragging action. The SHIFT key need not be held during the drag process.
- *Circle* – the left mouse button and SHIFT key are pressed when the cursor is located at the centre of the circle. The mouse is dragged (ie left button pressed) to the circumference of the circle and the button released. A rubber band is drawn during the dragging action. The SHIFT key need not be held during the drag process.

- *Polygon* – the SHIFT key is held down and the left mouse button is clicked (not held down), at each vertex of the polygon and the mouse moved to the next vertex, where again the mouse is clicked. The polygon is completed by clicking the mouse very close to the starting vertex (care needs to be taken to make this as accurate as possible). The ESCAPE key can be pressed at any time during this action in order to abort the process. A rubber band is drawn during the movement of the mouse.
- *Freehand* – the left mouse button and SHIFT key are pressed. A freehand figure is drawn as the mouse is dragged. When the mouse button is released the figure is completed with a straight line drawn from the final point to the initial point. The SHIFT key need not be held during the drag process.

In many applications regions can be selected by double clicking inside the region (or very close to the region in the case of a point or line) that is to be selected. Note that *inside* means inside the figure, independent of whether the region is an internal or external region. Once a region is selected its outline colour brightens. Many regions can be selected at the same time using the same process. Regions are unselected in the same way.

The user can manipulate selected regions. Selected regions are moved by holding down the CTRL key and using the four arrow keys. They can be resized by holding the ALT key and using the UP and DOWN arrow keys. For these manipulations it is important that the containing view has the mouse and keyboard focus. These are obtained by clicking the left mouse button on the view.

Regions can be stored on disk in *.rgn* files and loaded later when need.

Regions are used in the following places.

Image sequence image view

This view provides a convenient location for defining and saving regions (see section 3.2.3).

Overwrite regions filter

The overwrite regions filter provides the user with a mechanism for altering the pixel information in selected regions within each image. Regions are used to specify the regions that are to be manipulated.

Intensity field time series creation

Regions can be used to exclude certain regions from the calculation of the grid-based intensity field. For example a physical space within a fluid flow, that is not actually part of the flow, can be excluded from the production of the time series. The flow around a body is an example of such a situation.

Field views

Regions are used in a number of ways in the graphical views of scalar and vector fields (see chapter 10). The most common use is for defining a zoom rectangle that allows the user to enlarge a portion of the view. The selection, movement and enlargement of zoom rectangles are not supported in these views.

5.2 Views

Regions do not have any views. Their geometry can be seen by loading them into a view that supports their display, such as a image sequence image view.

5.3 Popup menu

Regions have no popup menus in the *ImageStream* application.

6 Filters

6.1 Description

This chapter summarises the various filters that are available to the user in *ImageStream*. These filters have also been discussed in some detail in the **System Theory and Design** Manual.

Filters are at the heart of the image processing function in *ImageStream*. A filter corresponds to an algorithm that takes the pixel intensity information of an image, transforms it pixel by pixel, and produces a new image based on the new pixel data.

Each filter has defined input and output data types. The pixel data of the input image must correspond to the expected input data type, and the pixel of the output image will correspond to the output data type. Currently 4 data types are supported. RGB colour, 10 bit gray-scale, 12 bit gray-scale, and real floating point values.

Filters are applied to image sequences as part of a filter pipeline, see chapter 3.

6.2 Views

All filters provide a details view.

6.2.1 Details View

Description

The details view displays the filter's parameters. These parameters are set in the dialog box when the filter is defined. See the appropriate section below for more information on a particular filter.

Access

A filter details view can be accessed by double clicking the filter object when it is displayed in the list box in the image sequence filter view, or through its popup menu.

6.3 Popup menu

All filters have a popup menu. This popup menu has the same form for all filters with a couple of exceptions that will be discussed below.

- *Save as…* – allows the user to save the filter in a disk file. The filter can be reloaded from disk in the image sequence filter view. Filter files have a *.flt* extension.
- *Open details view* – opens the details view of the filter.
- *Set parameters* – allows the user to set the various filter parameters. For each filter a dialog box is displayed that will prompt the user for the parameters. These parameters will be discussed in the following sections under each particular filter.

Two filters, the Barrel pincushion filter and the Polynomial field calibration filter, also include the following popup menu item.

- *Open analysis view* – opens the analysis view of the filter. This view typically allows the user to explore the filter, or its effects when applied to an image, in more detail.

And the Polynomial field calibration filter also includes the following popup menu item.

- *Calibrate* – this command instructs the filter to perform its calibration. This means generating calibration curves for all of the pixel blocks. This process can take 10s of seconds.

6.4 Amplify filter

An amplify filter scales the intensity at a pixel by a given factor. If the resulting intensity is outside the allowable range for that particular data type then the intensity will be set to the upper or lower limit as appropriate. For example if the red intensity of an RGB image is increased to a value greater than 255 by an amplify filter then the red intensity is set to 255.

Input data type: Any

Output data type: Same as input

Parameters

- *Filter name* – a user specified name for the filter. The default name is the filter type.
- *Image type* – the data type of the filter.
- *Red factor* – the scale factor for the red intensity. Only available when the data type is RGB.
- *Green factor* – the scale factor for the green intensity. Only available when the data type is RGB.
- *Blue factor* – the scale factor for the blue intensity. Only available when the data type is RGB.
- *Gray scale 10 bit factor* – the scale factor for the 10 bit gray scale intensity. Only available when the data type is 10 bit gray scale.
- *Gray scale 12 bit factor* – the scale factor for the 12 bit gray scale intensity. Only available when the data type is 12 bit gray scale.
- *Real factor* – the scale factor for the real value stored at a pixel. Only available when the data type is real value.

6.5 Absorption colour filter

The absorption colour filter calculates a real value that is proportional to the optical thickness of a fluid body by determining the attenuation of a particular light component as it passes through the fluid body. The fluid body is normally dyed in some way.

The calculation uses equation (5.1)

$$d(x,y) = -\ln\left(\frac{I(x,y)}{I_0(x,y)}\right) \qquad (6.1)$$

where $d(x,y)$ is the calculated optical thickness (actually proportional to it), $I(x,y)$ is the intensity at a pixel, while $I_0(x,y)$ is a reference intensity at the same pixel. Here, the intensity stands for the intensity of the red, green, blue gun, or the gray scale intensity.

The reference intensity can be obtained in two ways, either using a reference image (see section 6.6) or using the light intensity of another light component. The **System Theory and Design** manual provides more details of this process.

Input data type: RGB

Output data type: Real

Parameters

- *Filter name* – a user specified name for the filter. The default name is the filter type.
- *Absorption colour* – this drop down box allows the user to select the colour component that is being attenuated, and the colour that is being used as a reference intensity. All size possible combinations are available – eg red absorption relative to green, blue absorption relative to red etc.

6.6 Absorption image filter

This is the second of the absorption filters. This one utilises a reference image to provide the unattenuated reference pixel intensities. The size of the image used must match the size of the images in the image sequence. This is checked by the system.

Input data type: Any

Output data type: Real

Parameters

- *Filter name* – a user specified name for the filter. The default name is the filter type.
- *Absorption colour* – this drop down box allows the user to select the colour component that is being attenuated. This can be any component of an RGB signal, a 10 bit or 12 gray scale intensity or a real value. In addition it is possible to consider the attenuation of the ratio of the red and green, or red and blue, guns in an RGB image.
- *Image file* – the file that contains the reference image.
- *Browse* – this button allows the user to browse the directory structure in order to locate the reference image.

6.7 Barrel pincushion distortion filter

The barrel pincushion filter provides an algorithm that attempts to remove the lens distortion from an image. Barrel pincushion distortion is the most common type of distortion suffered by poor quality or wide angle lenses. Details of the transformation can be found in the **System Theory and Design** manual, and the equation used is repeated here in equation (6.2).

$$\mathbf{r}_s = \left(c_0 + c_1 r + c_2 r^2 + c_3 r^3\right)\mathbf{r}_d \qquad (6.2)$$

where $\mathbf{r}_s$ is the pixel location in the source (distorted image), $\mathbf{r}_d$ is the pixel location in the destination (undistorted image), r is the magnitude of $\mathbf{r}_d$, and the c_i are coefficients chosen by the user. It should be noted that this transformation depends on the location of the origin used. Generally the origin is taken as the physical centre of the image but the user is at liberty to change this.

Input data type: Any

Output data type: Same as input

Parameters

- *Filter name* – a user specified name for the filter. The default name is the filter type.
- *x centre (mm)* – the x coordinate of the origin used in the transformation. This will default to the centre of the image, but circumstances may arise where the centre is not the appropriate origin. For example an image sequence that has been trimmed unevenly using the Trimmed transformation would use an origin that is not centred on the trimmed image.
- *y centre (mm)* – the y coordinate of the origin used in the transformation. This will default to the centre of the image, but circumstances may arise where the centre is not the appropriate origin. For example an image sequence that has been trimmed unevenly using the Trimmed transformation would use an origin that is not centred on the trimmed image.
- *Coeff 0* – c0 in equation 6.2. This coefficient provides a general scaling of the image without affecting the distortion.
- *Coeff 1* – c1 in equation 6.2.
- *Coeff 2* – c2 in equation 6.2
- *Coeff 3* – c3 in equation 6.2

6.8 Bayer colour filter

This filter takes the pixel data from a Bayer filter based CCD camera and converts it into a full 32 bit colour image. This is achieved by interpolating the colour information at each pixel from surrounding pixels.

Input data type: RGB

Output data type: RGB

Parameters

- *Filter name* – a user specified name for the filter. The default name is the filter type.
- *Red gain* – the relative gains on the three RGB colour signals must be set in software. Effectively these gains set the white balance of the image and they are best obtained by taking an image of a white background and adjusting the gains until the three colour intensities match over the majority of the image. This field sets the red gain. In other words the red intensity will be amplified by this factor.
- *Green gain* – this field sets the green gain. In other words the green intensity will be amplified by this factor.
- *Blue gain* – this field sets the blue gain. In other words the blue intensity will be amplified by this factor.
- *Bayer coding* – this combo box allows the user to select the orientation of the colour filter. The default filter (standard Bayer coding) assumes a red pixel in the top left corner of the image. The inverted Bayer coding corresponds to a blue pixel in the top left corner.

6.9 Convert type filter

The convert type filter converts between data types. It provides a useful link between other filters in the filter pipeline whose types are not consistent. The user has a wide range of conversions from which to choose.

Input data type: User selectable

Output data type: User selectable

Parameters

- *Filter name* – a user specified name for the filter. The default name is the filter type.
- *Conversion* – this drop down list box provides a list of conversions that can be performed by this filter. For example Red to real will convert the red intensity from an RGB image into a real value.

6.10 Extract filter

The extract filter extracts one of the colours at each pixel and sets the others to zero. The colour may be red, green, blue or gray. In the first three cases the intensities of the other two guns are set to zero, while in the final case all three guns are set to the average of their original intensities.

Input data type: RGB

Output data type: RGB

Parameters

- *Filter name* – a user specified name for the filter. The default name is the filter type.
- *Colour to extract* – this drop down box allows the user to choose the colour they wish to extract from the RGB signal. They can choose, red, green, blue or gray.

6.11 Identity filter

The identity filter is a "do-nothing" filter. It takes any type of data input and leaves the intensity at each pixel unchanged and of the same type as the input.

Input data type: Any

Output data type: Same as input

Parameters

- *Filter name* – a user specified name for the filter. The default name is the filter type.

6.12 Invert filter

An inversion filter inverts one or more colours in the original image. Inverting a colour means subtracting its intensity from its maximum value. Thus an invert filter that inverts the green gun in an RGB signal will subtract the green intensity from 255 while the red and blue intensities will be left unaltered. The invert all filter performs the inversion operation on all guns. Gray scale images can be processed in the same way. For 10 bit gray scale the intensities are subtracted from 1023, while for 12 bit gray scale they are subtracted from 4095.

Input data type: Any (except real)

Output data type: Same as input

Parameters

- *Filter name* – a user specified name for the filter. The default name is the filter type.
- *Colour to invert* – this drop down box allows the user to choose the colour they wish to invert. It can be all, red, green, or blue for an RGB image, or 10 bit gray scale or 12 bit gray scale. Real images cannot be processed by this filter.

6.13 Overwrite regions filter

The overwrite regions filter provides a mechanism for selectively overwriting the pixel information in an image. The areas of the image that are to be overwritten are defined by a list of regions specified by the user. These regions can be created in the image view of an image sequence.

Input data type: Any

Output data type: Same as input

Parameters

- *Filter name* – a user specified name for the filter. The default name is the filter type.
- *Filter choice* – the type of image data.
- *Red value* – for an RGB image any pixel that lies within the specified regions will have its red intensity replaced by this value.
- *Green value* – for an RGB image any pixel that lies within the specified regions will have its green intensity replaced by this value.
- *Blue value* – for an RGB image any pixel that lies within the specified regions will have its blue intensity replaced by this value.
- *Gray scale 10 bit value* – for a 10 bit gray-scale image any pixel that lies within the specified regions will have its intensity replaced by this value.
- *Gray scale 12 bit value* – for a 12 bit gray-scale image any pixel that lies within the specified regions will have its intensity replaced by this value.
- *Real value* – for a real value image any pixel that lies within the specified regions will have its intensity replaced by this value.
- *Regions to overwrite* – this list box displays the regions that will be overwritten by the value(s) defined above. These regions are created in the image view of an image sequence and saved on disk. They are loaded into this list box using the Load button. Selected regions can be deleted by selecting them in the list box and clicking the Delete button. It is important to remember that regions can be of two types – internal and external. In the first case the region is defined as the pixels inside the geometric shape. In the second case the region is defined as the pixels that lie outside the geometric shape.

6.14 Polynomial field calibration filter

The polynomial field calibration filter (PFCF) is designed to convert light intensities into field concentrations (typically dye concentrations). It does this by generating calibration curves, based on a series of calibration images, which can transform the light intensity at any pixel into a field concentration. The size of the calibration images used must match the size of the images in the image sequence. This is checked by the system.

The user can decide whether the calibration curves are produced pixel by pixel or whether the calibration data is averaged over some small region within each image in order to obtain a more global calibration curve. In one extreme the user could choose to generate a single calibration curve for the whole image.

The calibration curves are polynomials, of user selectable order. They are based a number of calibration images provided by the user. These calibration images are assumed to correspond to a constant field value. A least squares fit is used for the calculation of the calibration coefficients.

The PFCF can work with any type of image.

Input data type: Any

Output data type: Real values

Parameters

- *Filter name* – a user specified name for the filter. The default name is the filter type.
- *Horizontal block size (pix)* – the width of the pixel block that is used to generate a calibration curve. The default size is 1 pixel wide by 1 pixel high. In this case every pixel within the image has its own calibration curve based on the equivalent pixels in the calibration images. If the block is greater than one pixel then the calibration information over the whole block is averaged in order to obtain a single calibration curve for all pixels within the block.
- *Vertical block size (pix)* – the height of the pixel block that is used to generate a calibration curve. See above.

- *Intensity cutoff* – during the calibration process, if the calibration intensity at a particular pixel exceeds the intensity cutoff this pixel intensity does not contribute to the calibration.
- *Base calibration on* – this drop down box specifies the type of pixel data that is expected in the calibration and image sequence images. It can be RGB, gray-scale or real.
- *Calibration images* – this list box displays the calibration images for this filter. The user can add, insert before, delete and clear these images. When an image is added to the list the user specifies both the name of the file containing the image and also the field value corresponding to the intensities in the image.
- *Order of polynomial* – the order of polynomial used for each of the calibration curves. Clearly the number of calibration images must equal or exceed the order of the polynomial in order for the least squares fitting process to work.

6.14.1 Analysis View

Description

The PFCF filter provides an analysis view that enables the user to explore the calibration. The calibration curve for any pixel in the image can be viewed. Similarly the calibration coefficients can be viewed for the whole image.

This view cannot be displayed until the filter has been calibrated. This can be done by selecting the "*Calibrate*" option in the popup menu (see section 6.3).

Pixel Fit

This tab allows the user to inspect the calibration curve for any pixel in the calibration images.

- *x pixel* – the column (and x coordinate) of the pixel for which the calibration is required.
- *y pixel* – the row (and y coordinate) of the pixel for which the calibration is required. The rows start at the bottom of the image. Thus pixel (0,0) lies at the bottom left hand corner of the images.
- *Generate fit…*– pressing this button generates the calibration curve for the designated pixel. A field object will appear in the calculated fields list and the user can view this field by double clicking it.

Calibration coefficients

This tab allows the user to generate a two-dimensional field of calibration coefficient values based on a rectangular grid.

- *Coefficient* – the coefficient to be displayed. Note that the last item in this drop down box is the least squares rms error.
- *Origin x coord (mm)* – the *x* coordinate, in mm, of the left most grid point.
- *Origin y coord (mm)* – the *y* coordinate, in mm, of the bottom most grid point.
- *Interpolation pts x spacing (mm)* – the spacing, in mm, between grid points in the *x* direction
- *Interpolation pts y spacing (mm)* – the spacing, in mm, between grid points in the *y* direction
- *Number of x points* – the total number of grid points in the *x* direction. This value is inclusive of both end grid points. Therefore the right most grid point has an *x* coordinate of $dx*(N-1) + x_0$, where *dx* is the *x* increment, *N* is the total number of grid points in the *x* direction and x_0 is the origin *x* coord of the grid.
- *Number of y points* – the total number of grid points in the *y* direction. This value is inclusive of both end grid points. Therefore the top most grid point has a *y* coordinate of $dy*(N-1) + y_0$, where *dy* is the *y* increment, *N* is the total number of grid points in the *y* direction and y_0 is the origin *y* coord of the grid.
- *Generate coeff field*…– pressing this button generates the coefficient field. A field object will appear in the calculated fields list and the user can view this field by double clicking it.

Access

This view is accessed through the popup menu of the filter. The filter is displayed in an image sequence filter view.

6.15 Remove filter

The remove filter sets the intensity of one of the guns in an RGB image to zero, thus effectively removing that colour from the image. The user can select the gun that is to be eliminated.

Input data type: RGB

Output data type: RGB

Parameters

- *Filter name* – a user specified name for the filter. The default name is the filter type.
- *Colour to remove* – the colour to be removed from the image. Red, green and blue are the options.

6.16 Subtract colour filter

A subtract colour filter subtracts the intensity of one of the colour guns, or a combination of their intensities, from all three guns in an RGB image. Thus if the user selects to subtract the blue gun, the intensity of the blue gun is subtracted from the red and green guns and the blue gun's intensity is set to 0. If any of these operations lead to a value less than 0 then 0 is used in its place.

Input data type: RGB

Output data type: RGB

Parameters

- *Filter name* – a user specified name for the filter. The default name is the filter type.
- *Filter choice* – the colour to be subtracted. The user can choose red, green, blue, average of red and green, average of red and blue, average or green and blue, or an average of all three colours.

6.17 Subtract image filter

The subtract image filter uses a reference image file as part of the subtraction process. Either the intensities of the pixels in the reference image are subtracted from the corresponding pixel intensities in the frames in the image sequence or vice versa.

The size of the image used must match the size of the images in the image sequence. This is checked by the system.

Input data type: Any

Output data type: Same as input

Parameters

- *Filter name* – a user specified name for the filter. The default name is the filter type.

- *Image file* – the file containing the reference image used in the subtraction process. The *Browse* button allows the user to navigate the file system.
- *Subtraction order* – specifies whether the reference image is subtracted from each of the images in the image sequence or vice versa.
- *Image type* – the type of data stored in the images – RGB, 10 bit gray-scale, 12 gray-scale or real values.

6.18 Threshold filter

The threshold filter acts as a tool for removing pixels from an image – ie setting pixel intensities to zero. The criterion for setting a pixel intensity to zero is based on a threshold intensity value at the pixel. If the intensity of the pixel exceeds, or falls below, a given threshold value then the replacement occurs. The threshold can be based on a single RBG component, a gray-scale value or a real value. And in turn the replacement can be applied to a single RGB component, or the gray-scale or real value.

Input data type: Any

Output data type: Same as input

Parameters

- *Filter name* – a user specified name for the filter. The default name is the filter type.
- *Threshold* – the threshold value upon with the replacement criterion is based.
- *Remove intensities* – specifies whether values below or above the threshold are replaced.
- *Base threshold on* – the intensity to which the threshold refers. This can an average of all RGB components, or a single one (ie red, green, blue), or it can be 10 bit gray-scale, 12bit gray-scale or real value.
- *Apply to* – this option specifies which intensities are set to 0 if the threshold condition is met. This can be all guns for an RGB image or a single gun (red, green, or blue) or it can be the 0 bit gray-scale intensity, 12bit gray-scale intensity or real value.

7 Intensity Fields

7.1 Description

An **intensity field** represents a time series of two-dimensional intensity fields that are calculated from the images in an image sequence. These intensities can be calculated directly from the original images, or from the output of the filter pipeline applied to the sequence. Depending on the type of data that is available the user can select the intensity to be used. For example for an RGB image the intensity can be any of the three colour intensities, or an average of all three.

An intensity field can be created in a number of ways, using either an intensity field producer (see chapter 8) or one of the velocity field transformations discussed below (see section 7.4).

An intensity field includes a number of calculators that allow the user to explore the data contained within the time series. These calculators are described in detail in chapter 9. In essence they calculate fields, such as intensity fluctuation or intensity gradient, derived from the intensity field.

Once created, an intensity field object is displayed in the *ImageStream* intensity field list view.

7.2 Views

The intensity field provides two views. These are

- details view
- calculator view

7.2.1 Details view

Description

The details view gives information about the intensity field. This information includes its name and description, details about the spatial grid on which the intensity field is defined, the time step

between fields and the total number of fields in the time series, and a list of the transformations that have been applied to generate the intensity field.

Access

Access to the details view is through the popup menu of the intensity field object.

Menus

View Menu

- *Producer details…* – displays the details view of the object that produced the intensity field. See chapter 8.

7.2.2 Calculator view

Description

The calculator view is the default view. It lists all of the calculators available for the intensity field. Currently seven calculators are available and these are discussed in detail in chapter 9.

Access

Access to the calculator view is through the popup menu of the intensity field object.

7.3 Popup menu

The intensity field provides the following options in its popup menu.

- *Save as…* – saves the intensity field in a new file. All intensity field files have the extension *.int6*. The new file name is displayed next to the intensity field name in the *ImageStream* intensity field list view.
- *Save* – saves the intensity field to disk. If the intensity field currently has no associated file then the user is prompted for a file name. All intensity field files have the extension *.int6*.
- *Save as raw text…* – saves the intensity field in a text format file that is readable by later versions of *ImageStream*. The text file has a *.itx6* extension.
- *Change name…* – allows the user to change the name of the intensity field. This name change will be reflected in the *ImageStream* intensity field list view.

- *Edit description…* – allows the user to change the description of the intensity field.
- *Delete fields…* – this option allows the user to delete fields within the intensity field. Deletion must occur from the beginning or end of the time series, never from the middle. This action requires confirmation from the user.
- *Open details view* – opens the details view.
- *Open calculator view* – opens the calculator view. This is the default view.
- *Create transformed intensity field…* – this allows the user to create a new intensity field through applying a transformation to this intensity field. A number of transformations are available and these are described in section 7.4. The resulting intensity field will appear in the *ImageStream* intensity field list view.
- *Delete calculated fields…* – deletes all of the calculated fields that currently exist within the field calculators (see chapter 9). The calculators and their calculated fields are all stored with the intensity field on disk. To save space it may be convenient to delete all of the calculated fields before saving the intensity field to disk.
- *Close all views* – closes all open views for this object.
- *Close* – removes the intensity field from *ImageStream*. All views are closed. If the intensity field has unsaved data then the user is prompted to save this data before closure.

7.4 Creating transformed intensity fields

Sometimes a simple transformation applied to an intensity field can make it more useful. The popup menu for an intensity field provides an option that allows the user to select a transformation to apply to the field. Currently six transformations are available.

7.4.1 Linear transformation

Under this transformation the spatial, temporal and field variables undergo a linear transformation. This transformation is particularly useful for the non-dimensionalisation of the data.

The parameters are

- *Field scaling* – all intensities are multiplied by this factor.
- *Constant*– this constant is added to all intensity.
- *Spatial scaling* – the spatial coordinates are multiplied by this scale factor.
- *x offset* – the x coordinate has this offset added to it.
- *y offset* – the y coordinate has this offset added to it.
- *Temporal scaling* – the time coordinate is multiplied by this scale factor.
- *t offset* – the time coordinate has this offset added to it.

7.4.2 Swap x transformation

This transformation reflects the intensity field in a vertical line through the middle of the field. Scaling of both the intensities and spatial coordinates is possible.

The parameters are

- *Field scaling* – all intensities are multiplied by this factor.
- *Spatial scaling* – the spatial coordinates are multiplied by this scale factor.

7.4.3 Swap y transformation

This transformation reflects the intensity field in a horizontal line through the middle of the field. Scaling of both the intensities and spatial coordinates is possible.

The parameters are

- *Field scaling* – all intensities are multiplied by this factor.
- *Spatial scaling* – the spatial coordinates are multiplied by this scale factor.

7.4.4 Translate fields using constant velocity transformation

The intensity fields in the time series are translated in space using a constant velocity vector. This transformation is useful if the user wishes to convert the intensity data into a moving frame of reference.

The parameters are

- *Reference frame* – the field corresponding to this time will be used as the reference frame. This means that its spatial coordinates are invariant under the shift.
- *First frame* – the first frame to which the transformation is applied.
- *Last frame* – the last frame to which the transformation is applied.
- *x shift velocity (mm/s)* – the x component of the applied constant velocity field. This is equivalent to the camera moving to the left at this speed. Frames after the reference frame are shifted in the positive x direction when the shift velocity is positive, while those before are shifted to the left.
- *y shift velocity (mm/s)* – the y component of the applied constant velocity field. This is equivalent to the camera moving downwards at this speed. Frames after the reference frame are shifted in the positive y direction when the shift velocity is positive, while those before are shifted downwards.

7.4.5 Translate fields using file displacements transformation

The intensity fields in the time series are translated in space using displacement vectors stored in a text file. The number of vectors in the file (in which the x and y components are separated by spaces, as are the different vectors) should match the number of time frames in the intensity field. If there are less vectors than expected the last vector is applied to the remaining frames in the sequence. If there are more vectors than expected, the excess vectors are ignored.

The parameters are

- *File name* – the name of the file containing the displacement vectors.

7.4.6 Average fields in time transformation

This transformation takes groups of fields and averages them in time to produce a single field. The end result is an intensity field with a reduced number of fields and a larger time step.

The parameters are

- *Average over frames* – the number of frames grouped in the averaging process.

8 Intensity Field Producers

8.1 Description

Intensity field objects are created by **intensity field producers**, of which there are a number. The most common source of an intensity field is from the pixel information in the images (or filtered images) in an image sequence. However intensity fields can also be produced by combining previously created fields. This chapter describes these various production mechanisms.

The main focus of the chapter will be the **intensity field image sequence producer**. Details of this will be presented in the first few sections. A producer that combines previously created intensity fields will be discussed in section 8.5.

The intensity field image sequence producer extracts intensities from the images (or their filtered counterparts) in an image sequence. These intensities are evaluated on a rectangular grid specified by the user. The grid can be as fine as the pixel resolution of the original images, thus providing intensity information at every pixel, or it can be coarse, thus resolving only the intensities at a few pixels (those that correspond to the nodes in the grid). The resolution of the grid will be highly dependent on the information required by the user.

This producer will also generate intensity spatial derivative information on the grid. The user can choose whether to base these derivatives on the intensities at the grid nodes, using a second order finite difference approximation (where possible, near the edges and sometimes in other locations only a first order scheme is available), or on adjacent pixels, again employing a second order finite difference scheme where possible. The use of the second of these is highly reliant of extremely accurate pixel intensities, and should be used with caution.

The user also has a number of other options that control the generation of the intensity field. Firstly the user can select the images within the sequence that are to be analysed. This does not need to include all of the images – any contiguous subset is allowed. Secondly the user can define regions that are to be excluded from the analysis. These regions generally correspond to domains within the

image where spurious intensity information resides (for example a solid boundary from which light reflections occur). Regions are discussed in chapters 3 and 5. They can be created in the image sequence image view, stored to disk and later loaded for the purpose of producing intensity fields.

Finally the user must specify the intensity in which they are interested. For an RGB image they may choose to calculate the red intensity, green intensity, blue intensity or an average of all three.

Once all of this information is provided the intensity field image sequence producer can generate the intensity fields.

8.2 Views

The intensity field image sequence producer has a single view – the details view.

8.2.1 Details view

Description

The details view displays all of the information specified when the intensity field producer was created. This includes the derivative scheme, the grid spacing and size, excluded regions and the frames included in the process. The details view has much in common with the dialog box that initially prompts the user when the producer is created. This creation process is described in detail in section 8.4 and the reader is referred there for more information.

Access

This view is opened through the *View* menu in the intensity field details view.

8.3 Popup menu

The intensity field image sequence producer object is never directly accessible to the user once it has been created. Therefore it does not provide a popup menu.

8.4 Creation of an intensity field image sequence producer

An intensity field image sequence producer is created by selecting *Create intensity field…*from the popup menu of an image sequence. A dialog box is presented to the user that allows the various parameters to be set. These are described in detail below.

8.4.1 Name

The intensity field is given a name that is used to identity it in the *ImageStream* intensity field list view. The default name is the same as that of the image sequence from which it is generated. Note that this name is an internal name to *ImageStream*. The user can choose a different name for the file in which the intensity field is saved, although this is probably not a wise decision.

Parameters to be set are:

- *Intensity field name* – a string by which *ImageStream* will refer to the intensity field.

8.4.2 Field

The intensity field to be calculated is selected in this tab. The options available to the user will be determined by the type of images provided to the producer. If the *Apply filters* box is not ticked then the producer deals directly with the images in the image sequence whose data type is specified by the sequence itself. On the other hand if the pipeline is applied to the images before the intensities are calculated then the output type of the final filter in the pipeline will determine the data type of the images. Either way, if RGB images are being analysed the user can choose which intensity they wish calculated – red, green, blue or an average of the three. For real images and gray scale images only one intensity is available.

- *Calculate average intensity* – for an RGB image this calculates the average intensity of the pixel at the grid point.
- *Calculate red intensity* – for an RGB image this calculates the red intensity of the pixel at the grid point.
- *Calculate green intensity* – for an RGB image this calculates the green intensity of the pixel at the grid point.

- *Calculate blue intensity* – for an RGB image this calculates the blue intensity of the pixel at the grid point.
- *Calculate gray scale 10 bit intensity* – for a 10 bit gray scale image this calculates the intensity of the pixel at the grid point.
- *Calculate gray scale 12 bit intensity* – for a 12 bit gray scale image this calculates the intensity of the pixel at the grid point.
- *Calculate real value* – for a real value image this calculates the real value of the pixel at the grid point.
- *Apply filters* – if this check box is ticked then the filter pipeline is applied to images before the intensity field is calculated.

8.4.3 Grid

The intensities are calculated on a uniform rectangular grid defined over the image. This grid need not cover a complete frame.

The grid is defined in physical coordinates (mm) with starting points in the *x* and *y* directions, increments in each direction, and the total numbers of grid points in each direction.

Note that the last grid settings are remembered by the system when the dialog box is next opened.

Parameters to be set are:

- *Origin x coord (mm)* – the *x* coordinate, in mm, of the left most grid point.
- *Origin y coord (mm)* – the *y* coordinate, in mm, of the bottom most grid point.
- *Interpolation pts x spacing (mm)* – the spacing, in mm, between grid points in the *x* direction
- *Interpolation pts y spacing (mm)* – the spacing, in mm, between grid points in the *y* direction
- *Number of x points* – the total number of grid points in the *x* direction. This value is inclusive of both end grid points. Therefore the right most grid point has an *x* coordinate of $dx*(N-1) + x_0$, where *dx* is the *x* increment, *N* is the total number of grid points in the *x* direction and x_0 is the origin *x* coord of the grid.
- *Number of y points* – the total number of grid points in the *y* direction. This value is inclusive of both end grid points. Therefore the top most grid point has a *y* coordinate of $dy*(N-1) + y_0$, where *dy* is the *y* increment, *N* is the total number of grid points in the *y* direction and y_0 is the origin *y* coord of the grid.

The three buttons at the bottom of the tab provide the following options.

- *Load grid* – new grid settings can be loaded from a disk file. This file has a *.grd* extension.
- *Save grid* – the current grid settings can be saved to a disk file. This file has a *.grd* extension.
- *Default grid* – the default grid settings can be reinstated with this button. This grid has 51 points in the x direction and 41 in the y direction. It covers the entire frame.

8.4.4 Derivative

This tab specifies how the spatial derivatives are to be calculated. The options here are as follows:

- *Grid based* – if this radio button is checked the intensity derivatives are calculated using central finite differences (where possible and forward/or backward difference elsewhere) based on the nodal values of the intensity. This is the default.
- *Pixel based* – if this radio button is checked the intensity derivatives are calculated using central finite differences (where possible and forward/or backward difference elsewhere) based on the pixel values of the intensity. These derivatives can lead to large errors if there is noise in the intensity signal.

8.4.5 Frames

Not all frames need be included in the intensity calculations. The first and last frame to be included can be set in this tab.

8.4.6 Regions

In this final tab the user can choose to define regions that are excluded from the intensity field calculations. Any grid point that lies in one of the defined regions will have an undefined intensity.

The buttons at bottom of the tab perform the following functions

- *Load* – allows the user to load regions from a disk file. The file has a *.rgn* extension. Note that such a file may contain many regions.
- *Delete* – when this button is pressed any regions selected in the list box will be deleted.

8.5 Combined intensity field producers

At times it may be necessary to combine two or more intensity fields. For example an experiment that requires the recording of a large number of images may need to be repeated and the video records from each repetition combined. *ImageStream* provides a tool for concatenating two or more intensity fields to produce a single larger intensity field. This tool, which is effectively another intensity filed producer, is accessed through the *ImageStream Intensity* menu and is discussed below.

8.5.1 Concatenated intensity field

A concatenated intensity field is constructed by placing the frames of a number of intensity fields in sequence, hence producing a single, longer time series. For each field in the sequence the user can specify spatial offsets in the x and y directions. These offsets are in units of grid points so that the grids of the resulting fields still align with one another. These offsets are relative to the origin of the first field in the sequence.

The user is presented with a dialog box that provides the following options. Once the parameters are set the user presses OK to produce the concatenated field. This field is displayed in the *ImageStream* intensity field list view.

Name

- *Intensity field name* – the name of the concatenated intensity field.

Intensity Files

- *List box*… – this list box displays the intensity fields that are to be concatenated. The order matches that in the resulting field.
- *Add* – this button allows the user to add intensity field files to the list. In addition the user is prompted for the grid offsets. These offsets are displayed in the list box along with the intensity field file name.
- *Insert before* – this button allows the user to insert an intensity field file before a file selected in the list box.
- *Delete* – this button deletes any selected files in the list box.

9 Intensity Field Calculators

9.1 Description

Field calculators have already been mentioned in chapter 7. Each intensity field contains a number of calculators that allow the user to extract interesting information from the intensity field. Each calculator produces a three dimensional field derived from the original intensity field. For example the grad(I) calculator calculates the gradient of the intensity field.

The available calculators are shown in table 9.1.

Symbol	Description
I	the instantaneous intensity field
I'	the fluctuating intensity field. The time average intensity, calculated from the entire time series, is subtracted from I to generate this field.
I'I'	the intensity of the fluctuating intensity.
grad(I)	the gradient of the intensity field
|grad(I)|	the magnitude of the gradient of the intensity field
dI/dx	partial derivative of intensity with respect to x
dI/dy	partial derivative of intensity with respect to y

Table 9.1 The list of available density field calculators.

9.2 Views

Each calculator provides a single view known as the manager view.

9.2.1 Manager View

Description

The field calculator manager view allows the user to calculate any one of the 30 different projections that the field calculator provides. All field calculators have the same manager view, and therefore this view will be described in a generic way.

The field calculator generates a derived three-dimensional field from the intensity fields of the intensity field object for which it is defined. For example the grad(I) calculator can calculate a three-dimensional vector field representing the gradient of the intensity field, where the three dimensions are x, y and t.

The manager view allows the user to calculate a range of zero-, one- and two-dimensional projections of this three-dimensional field. For each variable the user can select

- to leave that variable as a variable in the calculated field, specifying the domain of interest for that variable
- to average over that variable, specifying the domain of that variable over which the average is to be calculated, or
- to choose a value for that variable, at which the field will be calculated.

In addition, the user can choose to view the field along a line in the xy plane. This line is known as a **cut**. The user specifies the end points of the cut and number of points that lie along the cut. In general the cut points will not coincide with grid points and the field is interpolated onto these points. The time variable may still be specified as variable, averaged or constant for a cut field. Thus the user has a total of *30* different options.

A set of radio buttons, text boxes and drop down boxes allow the user to select the projection of interest. Once these controls have been set the field is calculated when the *Calculate field* button is pressed. Once calculated the field is displayed in the list box labelled, *Calculated fields*. Each field is given a unique name and an icon that indicates the type of field it represents (scalar, or vector, 0, 1, 2 or 3 dimensional).

The fields are given names that indicate their genesis. The name of the intensity field is followed by the field calculator name, which in turn if followed by parentheses containing three groups of symbols, separated by commas, referring to the variables, *x*, *y* and *t* respectively (except in the case of a cut). These symbols are

- the variable name, if that variable was chosen to be variable, followed by its domain

- the variable name in angle brackets (<>) if that variable was averaged, followed by the limits for the averaging process
- the variable name, followed by its value, if that variable was fixed,
- for a cut the symbol "xy" is used followed by two ordered pairs specifying the start and end points of the cut.

One or more fields can be selected and deleted by clicking the *Delete field* button.

When an intensity field object is saved to disk all of its calculators and their calculated fields are saved with it. As these fields can be easily recalculated, if storage space is at a premium, the user may wish to delete all fields before saving the intensity field object.

Access

The manager view is opened by double clicking on the field calculator in the intensity field calculator view, or through the popup menu of the field calculator object.

9.3 Popup menu

A field calculator provides the following options in its popup menu.

- *Save to text file…* – saves the full 3 dimensional field in a comma (,) delimited text file. Each two-dimensional field in the time series is stored in sequence.
- *Open manager view* – opens the manager view.

10 Fields

10.1 Description

Fields are calculated by field calculators and are displayed in the field calculator's manager view. These fields contain scalar or vector data defined on some regular rectangular grid of up to 3 dimensions. While these fields are quite general the independent variables are normally assumed to be *x*, *y* and *t*, where *x* and *y* are the horizontal and vertical spatial coordinates and *t* is time. Eight different field types are provided. These are

- zero dimensional vector field (a single vector)
- one dimensional vector field
- two-dimensional vector field
- three-dimensional vector field
- zero dimensional scalar field (a single scalar)
- one-dimensional scalar field
- two-dimensional scalar field
- three-dimensional scalar field

10.2 Views

A total of 20 different field views are available. Each field object only provides views that are appropriate to it. For example a two-dimensional scalar field provides four views – a table view, an animated graph view, a multi graph view, and a contour view.

The views are

- scalar field 0D table view
- scalar field 1D table view
- scalar field 1D graph view
- scalar field 1D PDF view
- scalar field 1D spectral view
- scalar field 2D table view
- scalar field 2D contour view
- scalar field 2D animated graph view

- scalar field 2D multi graph view
- scalar field 3D animated contour view
- vector field 0D table view
- vector field 1D table view
- vector field 1D graph view
- vector field 1D vector view
- vector field 2D table view
- vector field 2D animated graph view
- vector field 2D particle track view
- vector field 2D vector view
- vector field 3D particle track view
- vector field 3D animated vector view

All of these views offer the user some degree of control over how the view is displayed. It is important to note that if these display settings are changed they are not saved when the display is closed. Instead all parameters are restored to their default values when the view is reopened.

The functionality and interfaces provided by the graphical views (all views other than the table views) have much in common. A brief overview of this commonality is given here.

The functionality provided by each graphical view is available to the user in a number of locations. Firstly the menu bar at the top of the view window provides a number of menus that directly control the view. Every view has at least 4 menus. These are the *File*, *View*, *Colour* and *Zoom* menus. Secondly, below the menu bar is a group of toolbars that provide the user with easy access to some of the functionality available in the menu bar, as well as some additional information. For example, most views display the current cursor location and perhaps the value of the displayed field at the cursor location. A third way to access the functionality of the view is through a popup menu that is displayed when the right mouse button is clicked anywhere inside the graphical display area. Generally this popup menu does not provide the full list of menu options available in the main menu bar. The popup menu is designed to provide the user with easy access to some of the common view functions.

Below the toolbars lies the display area in which the view presents its graphical display of the field data. When the display is resized the

toolbars can take up a significant proportion of the view space, and all views allow these controls to be hidden.

All graphical views provide a zoom capability that allows the user to enlarge a certain portion of the displayed data. The zoom region is defined by an internal rectangle region (see chapter 6) drawn in the standard way on the display area. For most views the only region that can be drawn is an internal rectangle so no drawing tool selection is provided. In addition, for these views the zoom rectangle cannot be resized or moved. Instead it must be redrawn. In some views a change in viewing region can be achieved by setting the upper and lower limits of the two axes in four textboxes displayed in one of the view's toolbars. Whatever method is chosen to change the portion of the data that is displayed on screen, the unzoom option restores the default viewing region.

The nomenclature for the variables in the views should be mentioned. The horizontal and vertical axes are generically referred to as the *x* and *y* axes. Many views have text boxes that control the axes ranges, and that display the current cursor location as the mouse moves over the view. In all cases these text boxes are labeled using the *x* and *y* convention. However, the axes labels themselves indicate the actual variables being plotted. The convention here is that the independent variables for the various fields are designated *x* (horizontal space), *y* (vertical space) and *t* (time). Variables associated with the field itself are denoted *z*, or *u* and *v* as the components of a vector.

10.2.1 Scalar Field 0D table view

Description

The table view of a 0D scalar field simply displays the field value and details of the *x*, *y* and *t* variables. It is the default view.

Access

The table view of a 0D scalar field can be accessed by double clicking the field in the field calculator manager view or through the popup menu of the field object.

Menus

View menu

Change field decimal places… allows the user to select the number of decimal places used to display the field value.

10.2.2 Scalar Field 1D table view

Description

The table view of a 1D scalar field provides a table of the independent variable and its corresponding field value. Details of the other variables, whether they were averaged or fixed are displayed at the top of the view.

Access

The table view of a 1D scalar field can be accessed through the popup menu of the field object.

Menus

View menu

- *Change field decimal places…* – allows the user to select the number of decimal places used to display the field value.
- *Change position/time decimal places…* – allows the user to select the number of decimal places used to display the independent variable.

10.2.3 Scalar Field 1D graph view

Description

The graph view of a 1D scalar field provides a conventional plot of the scalar field. It is the default view for a scalar field 1D object.

The user has considerable control over the form of the display, and additional functionality, such as curve fitting, is provided.

Three toolbars are provided at the top of the view.

Display toolbar

- *Points* – if this check box is selected the field data points are shown on the graph. Their default colour is green. This can be changed in the *Colour* menu.
- *Lines* – if this check box is selected lines are drawn between the data points on the graph. Their default colour is red. This can be changed in the *Colour* menu.
- *Grid* – if this check box is selected a grid of lines is drawn on the graph. Their default colour is white. This can be changed in the *Colour* menu.
- *Curve fit* – if this check box is selected any curve fit to the data is shown on the graph. The default colour for the curve fit is white. This can be changed in the *Colour* menu.
- *Swap axes* – if this check box is selected the axes are swapped on the graph.

x axis toolbar

- *x axis min* – sets the minimum x value displayed.
- *x axis* max – sets the maximum x value displayed.
- *x* – displays the x value of the cursor as it moves over the graph.

y axis toolbar

- *y axis min* – sets the minimum y value displayed.
- *y axis* max – sets the maximum y value displayed.
- *y* – displays the y value of the cursor as it moves over the graph.

Access

The graph view of a 1D scalar field can be accessed by double clicking the field in the field calculator manager view, or through the popup menu of the field object.

Menus

File menu

- *Save as image*… – enables the user to save the graph as an image file in JPEG, PNG or BMP format. The user can select the resolution of the image.

View menu

- *Show points* – when selected the data points are displayed.

- *Show lines* – when selected the lines connecting the data points are displayed.
- *Show grid* – when selected the grid is displayed.
- *Show curve fit* – when selected the current curve fit is displayed.
- *Swap axes* – when selected the axes are swapped.
- *Set number of divisions…* – allows the user to change the number of grid lines.
- *Change field decimal places…*– allows the user to select the number of decimal places used to display the field value.
- *Change position/time decimal places…*– allows the user to select the number of decimal places used to display the independent variable.
- *Hide toolbars*– when selected the toolbars are hidden.

Colour menu

- *Change points colour…* – allows the user to change the colour of the data points.
- *Change lines colour…* – allows the user to change the colour of the lines drawn between data points.
- *Change grid/axes colour…* – allows the user to change the colour of the grid lines and axes.
- *Change label colour…* – allows the user to change the colour of the labels.
- *Change background colour…* – allows the user to change the background colour.
- *Change curve fit colour…* – allows the user to change the colour of the curve fit lines.

Curve fitting menu

- *Create curve fit…* – allows the user to fit a curve to a subset of the data points. The curves available are linear, polynomial, Gaussian and power.
- *View fit details…* – displays details of the curve fit including the fit parameters and the error.

Zoom menu

- *Zoom* – if a zoom rectangle has been defined (drawn on the view) this command will cause this region to fill the entire display. Data outside this rectangle will be clipped.
- *Unzoom* – reverses any changes to the limits of the display region and restores the full field domain.
- *Clear zoom rectangle* – clears the zoom rectangle.

Popup menu

- *Show points* – see View menu.
- *Show lines* – see View menu.
- *Show grid* – when see View menu.
- *Show curve fit* – see View menu.
- *Swap axes* – see View menu.
- *Zoom* – see Zoom menu.
- *Unzoom* – see Zoom menu.
- Clear zoom rectangle – see Zoom menu.

10.2.4 Scalar Field 1D PDF view

Description

The PDF view of a 1D scalar field displays either the probability density function (PDF) or cumulative probability density function (cumulative PDF) of the field. The range of the field values is divided into a number of equally sized bins (21 is the default number of bins) and the number of occurrences of field values lying within each bin is determined. When these numbers are normalized by the total number of field values the PDF of the field is obtained. If the numbers in the bins are summed from the smallest value of the independent variable to the largest (ie the left most bin to the right most bin) the cumulative PDF is obtained. The cumulative PDF ranges from 0 to 1.

The PDF or cumulative PDF can be displayed using points and lines, and/or bars.

Three toolbars are provided at the top of the view.

Display toolbar

- *Cumulative PDF* – if this check box is selected the cumulative PDF is displayed on the graph. Otherwise the PDF is displayed.
- *Points* – if this check box is selected the data points (PDF or cumulative PDF) are shown on the graph. Their default colour is green. This can be changed in the *Colour* menu.
- *Lines* – if this check box is selected lines are drawn between the data points on the graph. Their default colour is red. This can be changed in the *Colour* menu.

- *Bars* – if this check box is selected the data is plotted as bars, where each bar has a width equal to the width of one of the bins. Their default colour is light blue. This can be changed in the *Colour* menu.
- *Grid* – if this check box is selected a grid of lines is drawn on the graph. Their default colour is white. This can be changed in the *Colour* menu.
- *Curve fit* – if this check box is selected any curve fit to the data is shown on the graph. The default colour for the curve fit is white. This can be changed in the *Colour* menu.

x axis toolbar

- *x axis min* – sets the minimum x value displayed.
- *x axis* max – sets the maximum x value displayed.
- *x* – displays the x value of the cursor as it moves over the graph.

y axis toolbar

- *y axis min* – sets the minimum y value displayed.
- *y axis* max – sets the maximum y value displayed.
- *y* – displays the y value of the cursor as it moves over the graph.

Access

The PDF view of a 1D scalar field can be accessed through the popup menu of the field object.

Menus

File menu

- *Save as image*… – enables the user to save the graph as an image file in JPEG, PNG or BMP format. The user can select the resolution of the image.
- *Save to text file*… – enables the user to save the data displayed in a comma (,) delimited text file (.TXT or .CSV).

View menu

- *Show cumulative PDF* – when selected the cumulative PDF is displayed. Otherwise the PDF is displayed.
- *Show points* – when selected the data points are displayed.
- *Show lines* – when selected the lines connecting the data points are displayed.

- *Show bars* – when selected the data is represented by bars.
- *Show grid* – when selected the grid is displayed.
- *Show curve fit* – when selected the current curve fit is displayed.
- *Set number of bins…* – allows the user to change the number of bins used in the PDF calculation.
- *Set number of divisions…* – allows the user to change the number of grid lines.
- *Change field decimal places…*– allows the user to select the number of decimal places used to display the field value.
- *Change PDF places…*– allows the user to select the number of decimal places used to display the PDF or cumulative PDF.
- *Hide toolbars*– when selected the toolbars are hidden.

Colour menu

- *Change points colour…* – allows the user to change the colour of the data points.
- *Change lines colour…* – allows the user to change the colour of the lines drawn between data points.
- *Change bar colour…* – allows the user to change the colour of the bars.
- *Change grid/axes colour…* – allows the user to change the colour of the grid lines and axes.
- *Change label colour…* – allows the user to change the colour of the labels.
- *Change background colour…* – allows the user to change the background colour.
- *Change curve fit colour…* – allows the user to change the colour of the curve fit lines.

Curve fitting menu

- *Create curve fit…* – allows the user to fit a curve to a subset of the data points. The curves available are linear, polynomial, Gaussian and power.
- *View fit details…* – displays details of the curve fit including the fit parameters and the error.

Zoom menu

- *Zoom* – if a zoom rectangle has been defined (drawn on the view) this command will cause this region to fill the entire display. Data outside this rectangle will be clipped.

- *Unzoom* – reverses any changes to the limits of the display region and restores the full field domain.
- *Clear zoom rectangle* – clears the zoom rectangle.

Popup menu

- *Show points* – see View menu.
- *Show lines* – see View menu.
- *Show grid* – when see View menu.
- *Show curve fit* – see View menu.
- *Swap axes* – see View menu.
- *Zoom* – see Zoom menu.
- *Unzoom* – see Zoom menu.
- Clear zoom rectangle – see Zoom menu.

10.2.5 Scalar Field 1D spectral view

Description

The spectral view of a 1D scalar field provides a plot of the power spectral density (PSD) of the scalar field. Both linear and logarithmic axis scales are provided as options.

The PSD is calculated using a fast Fourier Transform (FFT) which includes a number of windowing algorithms, segmentation (and averaging of the PSD) and the use of overlapping segments. FFTs are computationally most efficient when the number of data points in the series equals an integer power of 2. Most fields do not conform to this requirement, and the FFT used in this view provides the user with 2 options. The first is to truncate the data series, starting from the beginning, so that its length equals the highest power of two less than its original length. The second option is to pad the complete series with zeros so that its length equals the lowest power of 2 greater than the original series length. All of these FFT settings can be changed using the *FFT* menu.

Curve fitting is available in this view. Note that the curve fits, even when logarithmic axes are used to display the data, are done on the original data set.

Three toolbars are provided at the top of the view.

Display toolbar

- *Points* – if this check box is selected the field data points are shown on the graph. Their default colour is green. This can be changed in the *Colour* menu.
- *Lines* – if this check box is selected lines are drawn between the data points on the graph. Their default colour is red. This can be changed in the *Colour* menu.
- *Grid* – if this check box is selected a grid of lines is drawn on the graph. Their default colour is white. This can be changed in the *Colour* menu. When logarithmic scales are used the grid lines are only drawn at the powers of 10.
- *Curve fit* – if this check box is selected any curve fit to the data is shown on the graph. The default colour for the curve fit is white. This can be changed in the *Colour* menu.
- *Log scales* – if this check box is selected the logarithmic scales are used to display both the x and y axes. The look of logarithmic axes is somewhat different to linear axes in that powers on 10 are the only axis scales drawn and grid lines are only associated with these values. All of the usual functionality, such as zooming or changing axis scales with the axes toolbars, are available.

x axis toolbar

- *x axis min* – sets the minimum x value displayed.
- *x axis* max – sets the maximum x value displayed.
- *x* – displays the x value of the cursor as it moves over the graph.

y axis toolbar

- *y axis min* – sets the minimum y value displayed.
- *y axis* max – sets the maximum y value displayed.
- *y* – displays the y value of the cursor as it moves over the graph.

Access

The spectral view of a 1D scalar field can be accessed through the popup menu of the field object.

Menus

File menu

- *Save as image*… – enables the user to save the graph as an image file in JPEG, PNG or BMP format. The user can select the resolution of the image.
- *Save as text file*… – enables the user to save the PSD as to a comma (,) delimited text file. The user can select the number of significant figures used to save the data.

View menu

- *Show points* – when selected the data points are displayed.
- *Show lines* – when selected the lines connecting the data points are displayed.
- *Show grid* – when selected the grid is displayed.
- *Show curve fit* – when selected the current curve fit is displayed.
- *Use log scales* – when selected log scales are used for both axes.
- *Set number of divisions*… – allows the user to change the number of grid lines when linear axes scales are used. When log scales are displayed grid lines are only drawn at each power of 10.
- *Change power decimal places*…– allows the user to select the number of decimal places used to display the PSD values. When log scales are used this setting becomes the number of significant figures.
- *Change frequency decimal places*…– allows the user to select the number of decimal places used to display the independent variable. When log scales are used this setting becomes the number of significant figures.
- *Hide toolbars*– when selected the toolbars are hidden.

Colour menu

- *Change points colour*… – allows the user to change the colour of the data points.
- *Change lines colour*… – allows the user to change the colour of the lines drawn between data points.
- *Change grid/axes colour*… – allows the user to change the colour of the grid lines and axes.
- *Change label colour*… – allows the user to change the colour of the labels.

- *Change background colour…* – allows the user to change the background colour.
- *Change curve fit colour…* – allows the user to change the colour of the curve fit lines.

FFT menu

- *Set FFT parameters…* – this menu item displays a dialog box in which the user can change the parameters governing the FFT used by the view. Parameters include: whether the series should be truncated or padded with zeros (an non-editable text box at the top of the dialog box indicates the number of data points that will be used in the FFT given the current settings); which windowing algorithm should be employed (the user can select from a number of standard windowing algorithms including Hann, Hamming, Gaussian etc); the alpha parameter which is used in the Kaiser and Gaussian windowing algorithms; the number of segments that the series should be broken into (the default is 1, ie the whole series is used to calculate the PSD); and whether overlapping of segments (by half the segment length) should be used when the series is segmented.

Curve fitting menu

- *Create curve fit…* – allows the user to fit a curve to a subset of the data points. The curves available are linear, polynomial, Gaussian and power.
- *View fit details…* – displays details of the curve fit including the fit parameters and the error.

Zoom menu

- *Zoom* – if a zoom rectangle has been defined (drawn on the view) this command will cause this region to fill the entire display. Data outside this rectangle will be clipped.
- *Unzoom* – reverses any changes to the limits of the display region and restores the full field domain.
- *Clear zoom rectangle* – clears the zoom rectangle.

Popup menu

- *Show points* – see View menu.
- *Show lines* – see View menu.
- *Show grid* – when see View menu.
- *Show curve fit* – see View menu.
- *Use log scales* – see View menu.

- *Zoom* – see Zoom menu.
- *Unzoom* – see Zoom menu.
- Clear zoom rectangle – see Zoom menu.

10.2.6 Scalar Field 2D table view

Description

The table view of the 2D scalar field displays the field values in tabular form. *x* axis values label the columns, and *y* axis values the rows.

The data from the table can be easily selected and copied and pasted into a spreadsheet.

Access

The table view of a 2D scalar field can be accessed through the popup menu of the field object.

Menus

View menu

- *Change field decimal places…* – allows the user to select the number of decimal places used to display the field value.
- *Change position/time decimal places…* –allows the user to select the number of decimal places used to display the independent variable.

10.2.7 Scalar Field 2D contour view

Description

The contour view of the 2D scalar field provides a false colour contour plot of the field. It is also possible to overlay a previously saved 2D vector field onto the contour plot. This is the default view of scalar field 2D object.

There are three toolbars at the top of the view. These are:

Display toolbar

- *Colour cells* – if this check box is selected then the data is displayed using colour cells to indicate the field value on the two-dimensional domain. The colours used can be changed in the *Colour* menu.
- *Contour lines* – if this check box is selected colour contour lines are drawn on the graph. The colours and the contour values can be changed in the *Colour* menu.
- *Grid* – if this check box is selected the grid on which the field is defined is drawn on the graph. Their default colour is white. This can be changed in the *Colour* menu.
- *Overlay* – if this check box is selected any 2D vector field that has been loaded is displayed on the graph.
- *Swap axes* – if this check box the axes are swapped.
- *Fill view* – if this check box is selected the aspect ratio of the data is ignored and the graph is expanded to fill the entire view.

Cursor toolbar

- *x* – displays the x value of the cursor as it moves over the graph.
- *y* – displays the y value of the cursor as it moves over the graph.
- *z* – displays the z (field) value at the location of the cursor as it moves over the graph.

Scale toolbar

- *Min colour value* – this is the field value corresponding to the colour at the bottom of the legend. Initially this is set to be the smallest field value on the grid. However, the user may choose to change this value. If grid values smaller than the value specified exist, then these will be coloured white.
- *Max colour value* – this is the field value corresponding to the colour at the top of the legend. Initially this is set to be the largest field value on the grid. However, the user may choose to change this value. If grid values larger than the value specified exist, then these will be coloured white.

It is possible that the field is undefined in some regions of the grid, due to the fact that there is no data in this region and hence the field is undefined. In this case the cell colour will be black.

The user can draw lines on the plot. These are referred to as adornments. These lines can be useful for determining coordinates

for cut views of the time series field data. An adornment is drawn by holding the CTRL key while dragging the mouse.

Access

The contour view of a 2D scalar field can be accessed by double clicking the field in the field calculator manager view or through the popup menu of the field object.

Menus

File menu

- *Save to image*… – enables the user to save the graph as an image file in JPEG, PNG or BMP format. The user can select the resolution of the image.

View menu

- *Show colour cells* – when selected the data points are displayed.
- *Show contours* – when selected the lines connecting the data points are displayed.
- *Show grid* – when selected the grid is displayed.
- *Show overlay* – when selected the current curve fit is displayed.
- *Swap axes* – when selected the axes are swapped.
- *Fill view* – allows the user to change the number of grid lines.
- *Set grid refinement factor* – the colour displayed in each colour cell is determined by the field values at the corners of the cell. A more refined colour field can be obtained by producing a finer grid on which the field values are linearly interpolated. The grid refinement factor determines the degree of grid refinement. The number of refined cells per original cell is given by the square of this factor.
- *Set overlay vector scale factor* – the length of the vectors of the overlaid vector field can be scaled using this factor.
- *Change field decimal places*… – allows the user to select the number of decimal places used to display the field value.
- *Change position/time decimal places*… – allows the user to select the number of decimal places used to display the independent variable.
- *Hide toolbars*– when selected the toolbars are hidden.

Colour menu

- *Change grid colour*… – allows the user to change the colour of the grid lines.

- *Change axes colour…* – allows the user to change the colour of the axes.
- *Change label colour…* – allows the user to change the colour of the labels.
- *Change background colour…* – allows the user to change the background colour.
- *Change adornment colour…* – allows the user to change the colour of adornments drawn on the view.
- *Change overlay vector colour…* – allows the user to change the colour of the vectors in an overlay vector field.
- *Change contour colour…* – allows the user to change the colours used for the contour lines and colour cells. The colours corresponding to the maximum, ¾, ½, ¼ and minimum field values can be set in a dialog box. The colours between these are interpolated. Buttons allow the default colours and a grey scale range to be created automatically.
- *Change contour values…* – allows the user to select the values of the contours displayed on the plot. Default values are chosen by the system when the view is first opened. These are based on the maximum and minimum scalar values displayed in the view. The user can add and remove contour values as they wish.
- *Use black contour lines* – all contour lines will displayed in black. In this way they can contrast with the colour cells.
- *Set contour values to default* – when the user changes the maximum and minimum colour values the contour values do not change. To reset the contour values to lie between these maximum and minimum values the user can select this menu option.

Overlay menu

- *Load overlay vector field…* – allows the user to load a two-dimensional vector field from a disk file to overlay on this graph.
- *Clear overlay vector field…* – clears any overlay vector field from the view.

Adornments menu

- *Clear adornments* – clears all adornments drawn on the figure.

Zoom menu

- *Zoom* – if a zoom rectangle has been defined (drawn on the view) this command will cause this region to fill the entire display. Data outside this rectangle will be clipped.
- *Unzoom* – reverses any changes to the limits of the display region and restores the full field domain.
- *Clear zoom rectangle* – clears the zoom rectangle..

Popup menu

- *Show colour cells* – see view menu.
- *Show contours* – see view menu.
- *Show grid* – see view menu.
- *Show overlay* – see view menu.
- *Swap axes* – see view menu.
- *Fill view* – see view menu.
- *Zoom* – see Zoom menu.
- *Unzoom* – see Zoom menu.
- Clear zoom rectangle – see Zoom menu.

10.2.8 Scalar Field 2D animated graph view

Description

The animated graph view of a 2D scalar field provides a plot of a one dimensional slice of the field, similar to the 1D scalar field graph view. However, because of the extra dimension, which for convenience is assumed to be associated with time, this 1D view varies in time. Thus this view allows the user to animate the one dimensional plot. If, for example, the field corresponded to the free surface elevation of a wave field as a function of time and space, this view would provide an animated view of the wave motion.

However, the view is not limited to a display of time-varying data. Suppose the field being viewed is a function of two space dimensions, say x and y. In this case the view is a one dimensional slice through this field in, say, the x direction, and the animation will display the slice for different values of y. The user is at liberty to swap which axis is interpreted as the time or animation variable.

Four toolbars are provided at the top of the view.

Animation toolbar

The animation toolbar provides a number of animation controls which allow the user to control the animation. Most of these controls are represented with icons which have a clear meaning. They are from left to right

- *Current time* – this combo box displays the current value of the animating variable. This variable originally corresponds to the "x" axis of the field, but may be changed (see View menu). The user can select any value in this box to move to a different position in the field.
- *Go to next time* – steps the display forward one frame.
- *Go to previous time*– steps the display back one frame.
- *Play*– starts the animation forward in time.
- *Play in reverse* – starts the animation backward in time.
- *Pause* – pauses the animation.
- *Go to start* – resets the display to the first frame.
- *Go to end* – resets the display to the last frame.
- *Animation timing delay (ms)* – this text box allows the user to change the speed of the animation. The displayed timing delay is the minimum time between frames. The actual delay may be longer if the drawing overhead is high.

Display toolbar

- *Points* – if this check box is selected the field data points are shown on the graph. Their default colour is green. This can be changed in the *Colour* menu.
- *Lines* – if this check box is selected lines are drawn between the data points on the graph. Their default colour is red. This can be changed in the *Colour* menu.
- *Grid* – if this check box is selected a grid of lines is drawn on the graph. Their default colour is white. This can be changed in the *Colour* menu.
- *Swap axes* – if this check box is selected the axes are swapped on the graph.

x axis toolbar

- *x axis min* – sets the minimum x value displayed.
- *x axis* max – sets the maximum x value displayed.
- *x* – displays the x value of the cursor as it moves over the graph.

y axis toolbar

- *y axis min* – sets the minimum y value displayed.
- *y axis* max – sets the maximum y value displayed.
- *y* – displays the y value of the cursor as it moves over the graph.

Access

The animated graph view of a 2D scalar field can be accessed through the popup menu of the field object.

Menus

File menu

- *Save as image*… – enables the user to save the graph as an image file in JPEG, PNG or BMP format. The user can select the resolution of the image.
- *Save to image sequence*… – enables the user to save each graph as an image in an image sequence. Image file types supported are JPEG, PNG and BMP format. The user can select the resolution of the images. The name, specified by the user, has the number of the frame added as a suffix. It is recommended that the base name ends in an underscore (_) in order to separate the file name from the frame numbering

View menu

- *Show points* – when selected the data points are displayed.
- *Show lines* – when selected the lines connecting the data points are displayed.
- *Show grid* – when selected the grid is displayed.
- *Swap axes* – when selected the axes are swapped.
- *Set number of divisions*… – allows the user to change the number of grid lines.
- *Change field decimal places*…– allows the user to select the number of decimal places used to display the field value.
- *Change position/time decimal places*…– allows the user to select the number of decimal places used to display the independent variable.
- *Hide toolbars*– when selected the toolbars are hidden.

Colour menu

- *Change points colour…* – allows the user to change the colour of the data points.
- *Change lines colour…* – allows the user to change the colour of the lines drawn between data points.
- *Change grid/axes colour…* – allows the user to change the colour of the grid lines and axes.
- *Change label colour…* – allows the user to change the colour of the labels.
- *Change background colour…* – allows the user to change the background colour.

Animation menu

- *Go to next time* – steps the display forward one frame. This has the same effect as the equivalent button in the animation toolbar.
- *Go to previous time*– steps the display back one frame. This has the same effect as the equivalent button in the animation toolbar.
- *Play* – starts the animation forward in time. This has the same effect as the equivalent button in the animation toolbar.
- *Play in reverse* – starts the animation backward in time. This has the same effect as the equivalent button in the animation toolbar.
- *Pause* – pauses the animation. This has the same effect as the equivalent button in the animation toolbar.
- *Go to start* – resets the display to the first frame. This has the same effect as the equivalent button in the animation toolbar.
- *Go to end* – resets the display to the last frame. The user is prompted to confirm this option. This has the same effect as the equivalent button in the animation toolbar.
- *Swap animation axis* – this option allows the user to select which of the two independent variables will be treated as the animation variable. The default setting is for the x axis to be treated as time, so the field is plotted against the y variable. Selecting this menu item will cause the x and y variables to swap roles.

Zoom menu

- *Zoom* – if a zoom rectangle has been defined (drawn on the view) this command will cause this region to fill the entire display. Data outside this rectangle will be clipped.
- *Unzoom* – reverses any changes to the limits of the display region and restores the full field domain.
- *Clear zoom rectangle* – clears the zoom rectangle.

Popup menu

- *Go to next time* – see Animation menu.
- *Go to previous time*– see Animation menu.
- *Play*– see Animation menu.
- *Play in reverse* – see Animation menu.
- *Pause* – see Animation menu.
- *Go to start* – see Animation menu.
- *Go to end* – see Animation menu.
- *Swap animation axis* – see Animation menu.
- *Show points* – see View menu.
- *Show lines* – see View menu.
- *Show grid* – when see View menu.
- *Swap axes* – see View menu.
- *Zoom* – see Zoom menu.
- *Unzoom* – see Zoom menu.
- Clear zoom rectangle – see Zoom menu.

10.2.9 Scalar Field 2D multi graph view

Description

The multi graph view of a 2D scalar field provides multiple overlaid plots of one dimensional slices of the field, similar to the 1D scalar field graph view. Thus if the two independent variables are x and t the user can view the spatial dependence of the field for each time, or alternatively he or she can view the temporal dependence of the field at each position in space.

Each graph is displayed in a different colour based on the colour legend displayed to the right of the plot. The range of colours employed for this purpose can be changed by the user.

Three toolbars are provided at the top of the view.

Display toolbar

- *Points* – if this check box is selected the field data points are shown on the graph. Each curve is displayed in a different colour and the lines and points are displayed with this colour . Curve colours can be changed under the *Colour* menu.

- *Lines* – if this check box is selected lines are drawn between the data points for each graph displayed. Each curve has its own colour that can be changed in the *Colour* menu.
- *Grid* – if this check box is selected a grid of lines is drawn on the graph. Their default colour is white. This can be changed in the *Colour* menu.
- *Swap axes* – if this check box is selected the axes are swapped on the graph.

x axis toolbar

- *x axis min* – sets the minimum x value displayed.
- *x axis* max – sets the maximum x value displayed.
- *x* – displays the x value of the cursor as it moves over the graph.

y axis toolbar

- *y axis min* – sets the minimum y value displayed.
- *y axis* max – sets the maximum y value displayed.
- *y* – displays the y value of the cursor as it moves over the graph.

Access

The multi graph view of a 2D scalar field can be accessed through the popup menu of the field object.

Menus

File menu

- *Save as image…* – enables the user to save the graph as an image file in JPEG, PNG or BMP format. The user can select the resolution of the image.

View menu

- *Show points* – when selected the data points are displayed.
- *Show lines* – when selected the lines connecting the data points are displayed.
- *Show grid* – when selected the grid is displayed.
- *Swap axes* – when selected the axes are swapped.
- *Set number of divisions…* – allows the user to change the number of grid lines.

- *Change field decimal places…* – allows the user to select the number of decimal places used to display the field value.
- *Change position/time decimal places…* – allows the user to select the number of decimal places used to display the independent variable.
- *Set curves details…* – when the view is first opened a curve for each value of y is plotted. This menu option displays a dialog box which allows the user to select the y values for which curves are to be plotted. The first y value, the number of y values and the increment between them can be specified.
- *Swap independent variable…* – allows the user to change the independent variable plotted on the x axis. The default setting is the x variable. Thus the y variable acts as the label for each curve.
- *Hide toolbars* – when selected the toolbars are hidden.

Colour menu

- *Change curve colours…* – allows the user to change the range of colours used for plotting the curves.
- *Change grid/axes colour…* – allows the user to change the colour of the grid lines and axes.
- *Change label colour…* – allows the user to change the colour of the labels.
- *Change background colour…* – allows the user to change the background colour.

Zoom menu

- *Zoom* – if a zoom rectangle has been defined (drawn on the view) this command will cause this region to fill the entire display. Data outside this rectangle will be clipped.
- *Unzoom* – reverses any changes to the limits of the display region and restores the full field domain.
- *Clear zoom rectangle* – clears the zoom rectangle.

Popup menu

- *Show points* – see View menu.
- *Show lines* – see View menu.
- *Show grid* – when see View menu.
- *Swap axes* – see View menu.
- *Swap independent variable* – see View menu.
- *Zoom* – see Zoom menu.
- *Unzoom* – see Zoom menu.
- Clear zoom rectangle – see Zoom menu.

10.2.10 Scalar Field 3D animated contour view

Description

The animated contour view of the 3D scalar field provides a colour cell plot of the field for each time step in the series. It has much in common with the contour plot for a 2D scalar field, but now an animation capability is added.

Min colour value and *Max colour value* in the *Scale* toolbar now correspond to the lowest and highest field value on the grid throughout the time series. This ensures that the colour values do not change during animation. As for the 2D scalar field plot, the user can override these colour settings.

This is the default view of scalar field 3D object.

There are four toolbars at the top of the view. These are:

Animation toolbar

- *Current time* – this combo box displays the current value of time. The user can select any value in this box to move to a different position in the field.
- *Go to next time* – steps the display forward one frame.
- *Go to previous time*…– steps the display back one frame.
- *Play*– starts the animation forward in time.
- *Play in reverse* – starts the animation backward in time.
- *Pause* – pauses the animation.
- *Go to start* – resets the display to the first frame.
- *Go to end* – resets the display to the last frame.
- *Animation timing delay (ms)* – this text box allows the user to change the speed of the animation. The displayed timing delay is the minimum time between frames. The actual delay may be longer if the drawing overhead is high.

Display toolbar

- *Colour cells* – if this check box is selected then the data is displayed using colour cells to indicate the field value on the two-dimensional domain. The colours used can be changed in the *Colour* menu.

- *Contour lines* – if this check box is selected colour contour lines are drawn on the graph. The colours and the contour values can be changed in the *Colour* menu.
- *Grid* – if this check box is selected the grid on which the field is defined is drawn on the graph. Their default colour is white. This can be changed in the *Colour* menu.
- *Swap axes* – if this check box the axes are swapped.
- *Fill view* – if this check box is selected the aspect ratio of the data is ignored and the graph is expanded to fill the entire view.

Cursor toolbar

- *x* – displays the x value of the cursor as it moves over the graph.
- *y* – displays the y value of the cursor as it moves over the graph.
- *z* – displays the z (field) value at the location of the cursor as it moves over the graph.

Scale toolbar

- *Min colour value* – this is the field value corresponding to the colour at the bottom of the legend. Initially this is set to be the smallest field value on the grid. However, the user may choose to change this value. If grid values smaller than the value specified exist, then these will be coloured white.
- *Max colour value* – this is the field value corresponding to the colour at the top of the legend. Initially this is set to be the largest field value on the grid. However, the user may choose to change this value. If grid values larger than the value specified exist, then these will be coloured white.

It is possible that the field is undefined in some regions of the grid, due to the fact that there is no data in this region and hence the field is undefined. In this case the cell colour will be black.

The user can draw lines on the plot. These are referred to as adornments. These lines can be useful for determining coordinates for cut views of the time series field data. An adornment is drawn by holding the CTRL key while dragging the mouse.

Access

The animated contour view of a 3D scalar field can be accessed by double clicking the field in the field calculator manager view, or through the popup menu of the field object.

Menus

File menu

- *Save to image…* – enables the user to save the graph as an image file in JPEG, PNG or BMP format. The user can select the resolution of the image.
- *Save to image sequence…* – enables the user to save each 2D contour plot as an image in an image sequence. Image file types supported are JPEG, PNG and BMP format. The user can select the resolution of the images. The name, specified by the user, has the number of the frame added as a suffix. It is recommended that the base name ends in an underscore (_) in order to separate the file name from the frame numbering.

View menu

- *Show colour cells* – when selected the data points are displayed.
- *Show contours* – when selected the lines connecting the data points are displayed.
- *Show grid* – when selected the grid is displayed.
- *Swap axes* – when selected the axes are swapped.
- *Fill view* – allows the user to change the number of grid lines.
- *Set grid refinement facto…r* – the colour displayed in each colour cell is determined by the field values at the corners of the cell. A more refined colour field can be obtained by producing a finer grid on which the field values are linearly interpolated. The grid refinement factor determines the degree of grid refinement. The number of refined cells per original cell is given by the square of this factor.
- *Change field decimal places…* – allows the user to select the number of decimal places used to display the field value.
- *Change position/time decimal places…* – allows the user to select the number of decimal places used to display the independent variables.
- *Hide toolbars*– when selected the toolbars are hidden.

Colour menu

- *Change grid colour…* – allows the user to change the colour of the grid lines.
- *Change axes colour…* – allows the user to change the colour of the axes.
- *Change label colour…* – allows the user to change the colour of the labels.

- *Change background colour…* – allows the user to change the background colour.
- *Change adornment colour…* – allows the user to change the colour of adornments drawn on the view.
- *Change contour colour…* – allows the user to change the colours used for the contour lines and colour cells. The colours corresponding to the maximum, ¾, ½, ¼ and minimum field values can be set in a dialog box. The colours between these are interpolated. Buttons allow the default colours and a grey scale range to be created automatically.
- *Change contour values* – allows the user to select the values of the contours displayed on the plot. Default values are chosen by the system when the view is first opened. These are based on the maximum and minimum scalar values displayed in the view. The user can add and remove contour values as they wish.
- *Use black contour lines* – all contour lines will displayed in black. In this way they can contrast with the colour cells.
- *Set contour values to default* – when the user changes the maximum and minimum colour values the contour values do not change. To reset the contour values to lie between these maximum and minimum values the user can select this menu option.

Adornments menu

- *Clear adornments* – clears all adornments drawn on the figure.

Zoom menu

- *Zoom* – if a zoom rectangle has been defined (drawn on the view) this command will cause this region to fill the entire display. Data outside this rectangle will be clipped.
- *Unzoom* – reverses any changes to the limits of the display region and restores the full field domain.
- *Clear zoom rectangle* – clears the zoom rectangle.

Popup menu

- *Go to next time* – see Animation menu.
- *Go to previous time*– see Animation menu.
- *Play* – see Animation menu.
- *Play in reverse* – see Animation menu.
- *Pause* – see Animation menu.
- *Go to start* – see Animation menu.
- *Go to end* – see Animation menu.

- *Show colour cells* – see View menu.
- *Show contours* – see View menu.
- *Show grid* – see View menu.
- *Swap axes* – see View menu.
- *Fill view* – see View menu.
- *Zoom* – see Zoom menu.
- *Unzoom* – see Zoom menu.
- Clear zoom rectangle – see Zoom menu.

10.2.11 Vector Field 0D table view

Description

The 0D vector field table view displays the vector components, and details of the three coordinates, *x*, *y* and *t*.

Access

The table view of a 0D vector field can be accessed by double clicking the field in the field calculator manager view or through the popup menu of the field object.

Menus

View menu

- *Change field decimal places…* – allows the user to select the number of decimal places used to display the field value.

10.2.12 Vector Field 1D table view

Description

The 1D vector field table view tabulates the independent field values together with the vector field at those points. Details of the remaining two independent coordinates are given above the table.

Access

The table view of a 1D vector field can be accessed through the popup menu of the field object.

Menus

View menu

- *Change field decimal places…* allows the user to select the number of decimal places used to display the vector field values.
- *Change position/time decimal places…* allows the user to select the number of decimal places used to display the independent variable.

10.2.13 Vector Field 1D graph view

Description

The graph view of a 1D vector field provides a conventional plot of the two field components (denoted *u* and *v* for the *x* and *y* components respectively), and has much in common with the graph view of a 1D scalar field. It is the default view for a 1D vector field object.

Three toolbars are provided at the top of the view.

Display toolbar

- *u points* – if this check box is selected the u field data points are shown on the graph. Their default colour is green. This can be changed in the *Colour* menu.
- *u lines* – if this check box is selected lines are drawn between the u data points on the graph. Their default colour is red. This can be changed in the *Colour* menu.
- *v points* – if this check box is selected the v field data points are shown on the graph. Their default colour is blue. This can be changed in the *Colour* menu.
- *v lines* – if this check box is selected lines are drawn between the v data points on the graph. Their default colour is yellow. This can be changed in the *Colour* menu.
- *Grid* – if this check box is selected a grid of lines is drawn on the graph. Their default colour is white. This can be changed in the *Colour* menu.
- *Swap axes* – if this check box is selected the axes are swapped on the graph.

x axis toolbar

- *x axis min* – sets the minimum x value displayed.
- *x axis* max – sets the maximum x value displayed.
- *x* – displays the x value of the cursor as it moves over the graph.

y axis toolbar

- *y axis min* – sets the minimum y value displayed.
- *y axis* max – sets the maximum y value displayed.
- *y* – displays the y value of the cursor as it moves over the graph.

Access

The graph view of a 1D vector field can be accessed by double clicking the field in the field calculator manager view, or through the popup menu of the field object.

Menus

File menu

- *Save as image…* – enables the user to save the graph as an image file in JPEG, PNG or BMP format. The user can select the resolution of the image.

View menu

- *Show u points* – when selected the u data points are displayed.
- *Show u lines* – when selected the lines connecting the u data points are displayed.
- *Show v points* – when selected the v data points are displayed.
- *Show v lines* – when selected the lines connecting the v data points are displayed.
- *Show grid* – when selected the grid is displayed.
- *Show u curve fit* – when selected the current curve fit for the u data is displayed.
- *Show v curve fit* – when selected the current curve fit for the v data is displayed.
- *Swap axes* – when selected the axes are swapped.
- *Set number of divisions…* – allows the user to change the number of grid lines.

- *Change field decimal places…*– allows the user to select the number of decimal places used to display the field value.
- *Change position/time decimal places…*– allows the user to select the number of decimal places used to display the independent variable.
- *Hide toolbars*– when selected the toolbars are hidden.

Colour menu

- *Change u points colour…* – allows the user to change the colour of the u data points.
- *Change u lines colour…* – allows the user to change the colour of the lines drawn between the u data points.
- *Change v points colour…* – allows the user to change the colour of the v data points.
- *Change v lines colour…* – allows the user to change the colour of the lines drawn between the v data points.
- *Change grid/axes colour…* – allows the user to change the colour of the grid lines and axes.
- *Change label colour…* – allows the user to change the colour of the labels.
- *Change background colour…* – allows the user to change the background colour.
- *Change u curve fit colour…* – allows the user to change the colour of the u data curve fit lines.
- *Change v curve fit colour…* – allows the user to change the colour of the v data curve fit lines.

Curve fitting menu

- *Create curve fit…* – allows the user to fit a curve to a subset of the u or v data sets. The curves available are linear, polynomial, Gaussian and power.
- *View u fit details…* – displays details of the u curve fit including the fit parameters and the error.
- *View v fit details…* – displays details of the v curve fit including the fit parameters and the error.

Zoom menu

- *Zoom* – if a zoom rectangle has been defined (drawn on the view) this command will cause this region to fill the entire display. Data outside this rectangle will be clipped.
- *Unzoom* – reverses any changes to the limits of the display region and restores the full field domain.
- *Clear zoom rectangle* – clears the zoom rectangle.

Popup menu

- *Show u points* – see View menu.
- *Show u lines* – see View menu.
- *Show v points* – see View menu.
- *Show v lines* – see View menu.
- *Show grid* – when see View menu.
- *Show u curve fit* – see View menu.
- *Show v curve fit* – see View menu.
- *Swap axes* – see View menu.
- *Zoom* – see Zoom menu.
- *Unzoom* – see Zoom menu.
- Clear zoom rectangle – see Zoom menu.

10.2.14 Vector Field 1D vector view

Description

The 1D vector field vector view provides a visual representation of the vector field as a function of the independent variable. This variable is plotted on the *x*, or *y*, axis, and at each grid point the corresponding field vector is drawn. The other axis is given a scale that corresponds to the length of vectors. The default scale for this axis is chosen so the maximum value is 4 times the size of the longest vector. The scales for both axes are readily changed using the text boxes in the Axes toolbar. A colour legend, based on vector length, is displayed to the right of the graph.

There are four toolbars at the top of the view.

Display toolbar

- *Vectors* – if this check box is selected the field vectors are displayed. The colour of the vectors can be changed under the *Colour* menu.
- *Grid* – if this check box is selected the grid is superimposed over the vector field. The colour of the grid can be changed under the *Colour* menu. *Swap axes* – if this box is checked the vectors are drawn on the other axis.
- *Clip* – if this box is checked any vectors drawn outside the displayed region will be clipped. This is particularly useful if the view has been zoomed.
- *Swap axes* – if this box is checked the vectors are drawn on the other axis.

Axes toolbar

- *x axis min* – sets the minimum x value displayed.
- *x axis* max – sets the maximum x value displayed.
- *y axis min* – sets the minimum y value displayed.
- *y axis* max – sets the maximum y value displayed.

Cursor toolbar

- *x* – displays the value of the independent variable at the location of the cursor as it moves over the graph. Note the label for this field will reflect the actual name of the independent variable.
- *u* – displays the x component of the vector field at the location of the cursor as it moves over the graph.
- *v* – displays the y component of the vector field at the location of the cursor as it moves over the graph.

Scale toolbar

- *Min colour value* – the lowest field value displayed. Initially this is set to be the magnitude of the shortest vector on the grid. However, the user may choose to change this value. If vectors shorter than the value specified exist, then these will be coloured white.
- *Max colour value* – the highest field value displayed. Initially this is set to be the magnitude of the longest vector on the grid. However, the user may choose to change this value. If vectors longer than the value specified exist, then these will be coloured white.

Access

The vector view of a 1D vector field can be accessed through the popup menu of the field object.

Menus

File menu

- *Save to image…* – enables the user to save the graph as an image file in JPEG, PNG or BMP format. The user can select the resolution of the image.

View menu

- *Show vectors* – when selected the vectors are displayed. The colour of the vectors can be changed under the *Colour* menu.

- *Show grid* – when selected the grid is displayed.
- *Clip* vectors – when selected vectors drawn outside the display region will be clipped.
- *Swap axes…* – when selected the axes are swapped.
- *Set number of divisions…* – allows the user to change the number of grid lines.
- *Change field decimal places…*– allows the user to select the number of decimal places used to display the field value.
- *Change position/time decimal places…*– allows the user to select the number of decimal places used to display the independent variable.
- *Hide toolbars*– when selected the toolbars are hidden.

Colour menu

- *Change vector colour* – allows the user to change the colour of the vectors. The colours corresponding to the maximum, ¾, ½, ¼ and minimum vector magnitudes can be set in a dialog box. The colours between these are interpolated. Buttons allow the default colours and a grey scale range to be created automatically.
- *Change grid/axes colour* – allows the user to change the colour of the axes.
- *Change label colour* – allows the user to change the colour of the labels.
- *Change background colour…* – allows the user to change the background colour.

Zoom menu

- *Zoom* – if a zoom rectangle has been defined (drawn on the view) this command will cause this region to fill the entire display. Data outside this rectangle will be clipped.
- *Unzoom* – reverses any changes to the limits of the display region and restores the full field domain.
- *Clear zoom rectangle* – clears the zoom rectangle.

Popup menu

- *Show vectors* – see View menu.
- *Show grid* – see View menu.
- *Clip vectors* – see View menu.
- *Swap axes* – see View menu.
- *Zoom* – see Zoom menu.
- *Unzoom* – see Zoom menu.
- Clear zoom rectangle – see Zoom menu.

10.2.15 Vector Field 2D table view

Description

The 2D vector field table view displays the two-dimensional vector field in a scrollable table. The horizontal coordinate is used to label the columns of the table, and the vertical coordinate the rows. Information about the third variable is given in the view header.

Access

The table view of a 2D vector field can be accessed through the popup menu of the field object.

Menus

View menu

- *Change field decimal places…* – allows the user to select the number of decimal places used to display the vector field values.
- *Change position/time decimal places…* –allows the user to select the number of decimal places used to display the independent variable.

10.2.16 Vector Field 2D vector view

Description

The 2D vector field vector view provides a visual representation of the vector field by drawing the field vectors at the given grid points. A colour legend, based on vector length, is displayed to the right of the graph.

This is the default view for the 2D vector field.

There are three toolbars at the top of the view. These are:

Display toolbar

- *Vectors* – if this is selected the field vectors are displayed. The colours of the vectors can be changed under the *Colour* menu.
- *Tangents* – if this is selected curves tangent to the vector field are drawn. If the vector field is a velocity field then these curves are streamlines, based on a streamfunction. The streamfunction is

calculated using an iterative technique where a new value for ψ at a grid point is obtained from the velocities at the four neighbouring grid points. The tolerance used in this iterative scheme, and the maximum number of iterations allowed, are set in the *View* menu. If the flow is essentially two-dimensional, so that the divergence is everywhere close to zero, the streamfunction is accurate. If there is a gain or loss of mass in regions within the flow the streamfunction will give spurious results.

- *Grid* – if this check box is selected the grid on which the field is defined is drawn on the graph. Their default colour is white. This can be changed in the Colour menu.
- *Fill view* – if this check box is selected the aspect ratio of the grid is ignored and the graph is expanded to fill the view.

Cursor toolbar

- *x* – displays the x value of the cursor as it moves over the graph.
- *y* – displays the y value of the cursor as it moves over the graph.
- *u* – displays the x component of the velocity field at the location of the cursor as it moves over the graph.
- *v* – displays the y component of the velocity field at the location of the cursor as it moves over the graph.
- *psi* – displays the value of the streamfunction at the location of the cursor as it moves over the graph.

Scale toolbar

- *Min colour value* – the field value corresponding to the colour at the bottom of the legend. Initially this is set to be the magnitude of the shortest vector on the grid. However, the user may choose to change this value. If vectors shorter than the value specified exist, then these will be coloured white.
- *Max colour value* – the field value corresponding to the colour at the top of the legend. Initially this is set to be the magnitude of the longest vector on the grid. However, the user may choose to change this value. If vectors longer than the value specified exist, then these will be coloured white.

Access

The vector view of a 2D vector field can be accessed by double clicking the field in the field calculator manager view or through the popup menu of the field object.

Menus

File menu

- *Save to image…* – enables the user to save the graph as an image file in JPEG, PNG or BMP format. The user can select the resolution of the image.

View menu

- *Show vectors* – when selected the vectors are displayed. The colour of the vectors can be changed under the *Colour* menu.
- *Show tangents* – when selected the tangent field is displayed.
- *Show grid* – when selected the grid is displayed.
- *Fill view* – when selected the graph is expanded to fill the entire view.
- *Change contour values…* – allows the user to change the values of the streamfunction used to draw the tangents.
- *Set contour values to default…* – sets the streamfunction values, used for drawing the tangent lines, back to their default values.
- *Set vector scale factor…* – allows the user to scale the vectors as these vectors are initially drawn using an arbitrary scale.
- *Set vectors to display…* – by default all vectors defined on the grid are shown. For dense grids this can lead to a congested view. This option allows the user to exclude vectors from the display.
- *Clip vectors* – when selected, any vectors lying outside the displayed region will be clipped.
- *Set tangent line accuracy…* – sets the accuracy to within which the streamfunction should be calculated when determining the tangent lines.
- *Set tangent line iteration limit…* – sets the maximum number of iterations used to determine streamfunction. This limit is needed because convergence may not occur for data that is not two-dimensional, or nearly two-dimensional.
- *Change field decimal places…*– allows the user to select the number of decimal places used to display the field value.
- *Change position/time decimal places…*– allows the user to select the number of decimal places used to display the independent variable.
- *Hide toolbars*– when selected the toolbars are hidden.

Colour menu

- *Change grid colour…* – allows the user to change the colour of the grid lines.
- *Change axes colour…* – allows the user to change the colour of the axes.
- *Change label colour…* – allows the user to change the colour of the labels.
- *Change vector colour…* – allows the user to change the colour of the vectors. The colours corresponding to the maximum, ¾, ½, ¼ and minimum vector magnitudes can be set in a dialog box. The colours between these are interpolated. Buttons allow the default colours and a grey scale range to be created automatically.
- *Change tangent line colour…* – allows the user to change the colour of the tangent lines. By default these are white.
- *Change background colour…* – allows the user to change the background colour.
- *Change adornment colour…* – allows the user to change the colour of adornments drawn on the view.

Adornments menu

- *Clear adornments* – clears all adornments drawn on the graph.

Zoom menu

- *Zoom* – if a zoom rectangle has been defined (drawn on the view) this command will cause this region to fill the entire display. Data outside this rectangle will be clipped.
- *Unzoom* – reverses any changes to the limits of the display region and restores the full field domain.
- *Clear zoom rectangle* – clears the zoom rectangle.

Popup menu

- *Show vectors* – see View menu.
- *Show tangents* – see View menu.
- *Show grid* – see View menu.
- *Fill view* – see View menu.
- *Zoom* – see Zoom menu.
- *Unzoom* – see Zoom menu.
- Clear zoom rectangle – see Zoom menu.

10.2.17 Vector Field 2D animated graph view

Description

The animated graph view of the 2D vector field is similar to the animated graph view of a 2D scalar field, except now both vector components are plotted.

The view has four toolbars at the top of the view.

Animation toolbar

- *Current time* – this combo box displays the current value of the animating variable. This variable originally corresponds to the "x" axis of the field, but may be changed (see View menu). The user can select any value in this box to move to a different position in the field.
- *Go to next time* – steps the display forward one frame.
- *Go to previous time*– steps the display back one frame.
- *Play*– starts the animation forward in time.
- *Play in reverse* – starts the animation backward in time.
- *Pause* – pauses the animation.
- *Go to start* – resets the display to the first frame.
- *Go to end* – resets the display to the last frame.
- *Animation timing delay (ms)* – this text box allows the user to change the speed of the animation. The displayed timing delay is the minimum time between frames. The actual delay may be longer if the drawing overhead is high.

Display toolbar

- *u points* – if this check box is selected the u field data points are shown on the graph. Their default colour is green. This can be changed in the *Colour* menu.
- *u lines* – if this check box is selected lines are drawn between the u data points on the graph. Their default colour is red. This can be changed in the *Colour* menu.
- *v points* – if this check box is selected the v field data points are shown on the graph. Their default colour is blue. This can be changed in the *Colour* menu.
- *v lines* – if this check box is selected lines are drawn between the v data points on the graph. Their default colour is yellow. This can be changed in the *Colour* menu.

- *Grid* – if this check box is selected a grid of lines is drawn on the graph. Their default colour is white. This can be changed in the *Colour* menu.
- *Swap axes* – if this check box is selected the axes are swapped on the graph.

x axis toolbar

- *x axis min* – sets the minimum x value displayed.
- *x axis* max – sets the maximum x value displayed.
- *x* – displays the x value of the cursor as it moves over the graph.

y axis toolbar

- *y axis min* – sets the minimum y value displayed.
- *y axis* max – sets the maximum y value displayed.
- *y* – displays the y value of the cursor as it moves over the graph.

Access

The animated graph view of a 2D vector field can be accessed through the popup menu of the field object.

Menus

File menu

- *Save as image…* – enables the user to save the graph as an image file in JPEG, PNG or BMP format. The user can select the resolution of the image.
- *Save to image sequence…* – enables the user to save each graph as an image in an image sequence. Image file types supported are JPEG, PNG and BMP format. The user can select the resolution of the images. The name, specified by the user, has the number of the frame added as a suffix. It is recommended that the base name ends in an underscore (_) in order to separate the file name from the frame numbering

View menu

- *Show u points* – when selected the u data points are displayed.
- *Show u lines* – when selected the lines connecting the u data points are displayed.
- *Show v points* – when selected the v data points are displayed.

- *Show v lines* – when selected the lines connecting the v data points are displayed.
- *Show grid* – when selected the grid is displayed.
- *Swap axes* – when selected the axes are swapped.
- *Set number of divisions…* – allows the user to change the number of grid lines.
- *Change field decimal places…* – allows the user to select the number of decimal places used to display the field value.
- *Change position/time decimal places…* – allows the user to select the number of decimal places used to display the independent variable.
- *Hide toolbars*– when selected the toolbars are hidden.

Colour menu

- *Change u points colour…* – allows the user to change the colour of the u data points.
- *Change u lines colour…* – allows the user to change the colour of the lines drawn between u data points.
- *Change v points colour…* – allows the user to change the colour of the v data points.
- *Change v lines colour…* – allows the user to change the colour of the lines drawn between v data points.
- *Change grid/axes colour…* – allows the user to change the colour of the grid lines and axes.
- *Change label colour…* – allows the user to change the colour of the labels.
- *Change background colour…* – allows the user to change the background colour.

Animation menu

- *Go to next time* – steps the display forward one frame. This has the same effect as the equivalent button in the animation toolbar.
- *Go to previous time*– steps the display back one frame. This has the same effect as the equivalent button in the animation toolbar.
- *Play* – starts the animation forward in time. This has the same effect as the equivalent button in the animation toolbar.
- *Play in reverse* – starts the animation backward in time. This has the same effect as the equivalent button in the animation toolbar.
- *Pause* – pauses the animation. This has the same effect as the equivalent button in the animation toolbar.
- *Go to start* – resets the display to the first frame. This has the same effect as the equivalent button in the animation toolbar.

- *Go to end* – resets the display to the last frame. The user is prompted to confirm this option. This has the same effect as the equivalent button in the animation toolbar.
- *Swap animation axis* – this option allows the user to select which of the two independent variables will be treated as the animation variable. The default setting is for the x axis to be treated as time, so the field is plotted against the y variable. Selecting this menu item will cause the x and y variables to swap roles.

Zoom menu

- *Zoom* – if a zoom rectangle has been defined (drawn on the view) this command will cause this region to fill the entire display. Data outside this rectangle will be clipped.
- *Unzoom* – reverses any changes to the limits of the display region and restores the full field domain.
- *Clear zoom rectangle* – clears the zoom rectangle.

Popup menu

- *Go to next time* – see Animation menu.
- *Go to previous time*– see Animation menu.
- *Play*– see Animation menu.
- *Play in reverse* – see Animation menu.
- *Pause* – see Animation menu.
- *Go to start* – see Animation menu.
- *Go to end* – see Animation menu.
- *Swap animation axis* – see Animation menu.
- *Show u points* – see View menu.
- *Show u lines* – see View menu.
- *Show v points* – see View menu.
- *Show v lines* – see View menu.
- *Show grid* – when see View menu.
- *Swap axes* – see View menu.
- *Zoom* – see Zoom menu.
- *Unzoom* – see Zoom menu.
- Clear zoom rectangle – see Zoom menu.

10.2.18 Vector Field 2D particle track view

Description

The particle track view of the 2D vector field is primarily suited to vector fields corresponding to fluid velocity fields. It is designed to enable the user to view the fluid motion by seeding the flow with particles and tracking their trajectories. The flow is assumed steady, so that the velocity field is invariant in time at each point in the flow. This seeding process is achieved with an object known as a **particle seeder** – see chapter 16. As the field does not possess any variation in time a simulation time is defined by the user. This includes a time step between frames and the number of frames in the simulation. These two parameters are set to 0.04 and 500 by default.

The underlying display in this view is almost identical to the 2D vector field vector view. One important difference worth noting is that this view allows the user to create a full suite of regions. These are important in defining the zones in which the particle seeders introduce the particles to the flow. Therefore the zooming capability of this view is somewhat different to the majority of the other views. It follows the same rules as used in the video clip graph view. Only internal rectangles can be used for zooming. If the zoom command is executed and only one internal rectangle is defined, then this rectangle is used as the zoom rectangle. If more than one is defined then the user is presented with a dialog box from which they can choose the zoom rectangle. Another approach is to double click the left mouse button, while holding down the SHIFT key, inside the rectangle that is to be used as the zoom rectangle.

There are three toolbars at the top of the view. These are:

Animation toolbar

- *Current time* – this combo box displays the current value of time. The user can select any value in this box to move to a different time.
- *Go to next time* – steps the display forward one frame.
- *Go to previous time*– steps the display back one frame.
- *Play*– starts the animation forward in time.
- *Play in reverse* – starts the animation backward in time.
- *Pause* – pauses the animation.
- *Go to start* – resets the display to the first frame.

- *Go to end* – resets the display to the last frame.
- *Animation timing delay (ms)* – this text box allows the user to change the speed of the animation. The displayed timing delay is the minimum time between frames. The actual delay may be longer if the drawing overhead is high.

Display toolbar

- *Particles* – if this is selected the seeded particles are displayed. The colour of the particles can be changed under the *Colour* menu.
- *Vectors* – if this is selected the field vectors are displayed. The colours of the vectors can be changed under the *Colour* menu.
- *Grid* – if this check box is selected the grid on which the field is defined is drawn on the graph. Their default colour is white. This can be changed in the Colour menu.
- *Fill view* – if this check box is selected the aspect ratio of the grid is ignored and the graph is expanded to fill the view.
- *Dimmer* – this slide control allows the user dim or brighten the vector field which acts as a backdrop to this view. When the vector field is displayed at full brightness it can be difficult to clearly see the motion of the particles, and hence dimming the field can be useful.

Cursor toolbar

- *x* – displays the x value of the cursor as it moves over the graph.
- *y* – displays the y value of the cursor as it moves over the graph.
- *u* – displays the x component of the velocity field at the location of the cursor as it moves over the graph.
- *v* – displays the y component of the velocity field at the location of the cursor as it moves over the graph.
- *Drawing tool* – this text field displays the current drawing tool. The drawing tool is set in the *Region* menu.

Access

The particle track view of a 2D vector field can be accessed through the popup menu of the field object.

Menus

File menu

- *Save as image…* – enables the user to save the figure as an image file in JPEG, PNG or BMP format. The user can select the resolution of the image.

- *Save to image sequence…* – enables the user to save each figure in the animation as an image in an image sequence. Image file types supported are JPEG, PNG and BMP format. The user can select the resolution of the images. The name, specified by the user, has the number of the frame added as a suffix. It is recommended that the base name ends in an underscore (_) in order to separate the file name from the frame numbering.
- *Save tracks to text file…* – enables the user to save the particle tracks to a comma delimited text file (.TXT or .CSV). The x coordinates of track of the first particle are stored first, followed by the y coordinates of the first particle, then the x coordinates of the second and so on.

View menu

- *Show particles* – when selected the particles are displayed. The colour of the particles can be changed under the *Colour* menu.
- *Show vectors* – when selected the vectors are displayed. The colour of the vectors can be changed under the *Colour* menu.
- *Show grid* – when selected the grid is displayed.
- *Fill view* – when selected the graph is expanded to fill the entire view.
- *Set vector scale factor…* – allows the user to scale the vectors as these vectors are initially drawn using an arbitrary scale.
- *Set vectors to display…* – by default all vectors defined on the grid are shown. For dense grids this can lead to a congested view. This option allows the user to exclude vectors from the display.
- *Clip vectors* – when selected, any vectors lying outside the displayed region will be clipped.
- *Change field decimal places…*– allows the user to select the number of decimal places used to display the field value.
- *Change position/time decimal places…*– allows the user to select the number of decimal places used to display the independent variable.
- *Hide toolbars*– when selected the toolbars are hidden.

Colour menu

- *Change vector colour…* – allows the user to change the colour of the vectors. The colours corresponding to the maximum, ¾, ½, ¼ and minimum vector magnitudes can be set in a dialog box. The colours between these are interpolated. Buttons allow the default colours and a grey scale range to be created automatically.
- *Change grid colour…* – allows the user to change the colour of the grid lines.
- *Change axes colour…* – allows the user to change the colour of the axes.

- *Change label colour…* – allows the user to change the colour of the labels.
- *Change background colour…* – allows the user to change the background colour.

Animation menu

- *Set time step…* – as the field used in this particle tracking simulation is a single two dimensional field, the user is required to specify the time step between frames in the simulation and the number of frames in the simulation. This menu item allows the user to set the time step. The default is 0.04s.
- *Set number of frames…* – this item allows the user to specify the total number of frames used in the simulation. The default is 500.
- *Go to next time* – steps the display forward one frame. This has the same effect as the equivalent button in the animation toolbar.
- *Go to previous time*– steps the display back one frame. This has the same effect as the equivalent button in the animation toolbar.
- *Play* – starts the animation forward in time. This has the same effect as the equivalent button in the animation toolbar.
- *Play in reverse* – starts the animation backward in time. This has the same effect as the equivalent button in the animation toolbar.
- *Pause* – pauses the animation. This has the same effect as the equivalent button in the animation toolbar.
- *Go to start* – resets the display to the first frame. This has the same effect as the equivalent button in the animation toolbar.
- *Go to end* – resets the display to the last frame. The user is prompted to confirm this option. This has the same effect as the equivalent button in the animation toolbar.

Seeder Menu

- *Display particle seeders…* – displays all of the currently defined particle seeders in a list box. This list box allows the user to create, load and save particle seeders.
- *Load particle seeder…*– loads a previously saved particle seeder from disk.
- *Create particle seeder…*– creates a new particle seeder. The user is given a dialog box in which to enter the particle seeder details (see chapter 16).
- *Generate tracks* – all current particle tracks are deleted and new particle tracks are created. When this item is selected the system creates all of the particle tracks throughout the throughout the time period of the simulation.

- *Delete tracks…* – deletes all particle tracks. The user is prompted to confirm this option.

Region Menu

- *Display regions…* – displays all of the currently defined regions in a list box.
- *Load regions…*– previously stored regions are loaded from a disk file and added to the currently defined regions. Regions are stored in a file with a *.rgn* extension.
- *Save regions as…*– saves all of the currently defined regions in a *.rgn* disk file.
- *Delete all regions* – deletes all currently defined regions. The user is prompted to confirm this option.
- *Delete selected regions* – deletes all selected regions.
- *Select all regions* – selects all regions.
- *Select <-> unselect* – makes all selected regions unselected and vice versa.
- *Invert selected regions* – converts all selected regions that are interior regions into exterior regions and vice versa.
- *Invert all regions* – converts all regions that are interior regions into exterior regions and vice versa.
- *Draw point* – sets the current drawing tool to a point.
- *Draw line* – sets the current drawing tool to a line.
- *Draw internal rectangle* – sets the current drawing tool to an internal rectangle. This is the default.
- *Draw internal circle* – sets the current drawing tool to an internal circle.
- *Draw internal polygon* – sets the current drawing tool to an internal polygon.
- *Draw internal freehand* – sets the current drawing tool to an internal freehand.
- *Draw external rectangle* – sets the current drawing tool to an external rectangle.
- *Draw external circle* – sets the current drawing tool to an external circle.
- *Draw external polygon* – sets the current drawing tool to an external polygon.
- *Draw external freehand* – sets the current drawing tool to an external freehand.

Zoom Menu

- *Zoom…* – causes the view to be zoomed. If only one internal rectangle is currently defined then it is used as the zoom rectangle. If more than one internal rectangle exists then a dialog box is presented to the user, listing all internal rectangles currently defined. The user can select a zoom rectangle from the list.
- *Unzoom…*– this undoes any zoom operation by redisplaying the entire field.

Popup menu

- *Go to next time* – see Animation menu.
- *Go to previous time*– see Animation menu.
- *Play*– see Animation menu.
- *Play in reverse* – see Animation menu.
- *Pause* – see Animation menu.
- *Go to start* – see Animation menu.
- *Go to end* – see Animation menu.
- *Show particles* – see View menu.
- *Show vectors* – see View menu.
- *Show grid* – see View menu.
- *Zoom* – see Zoom menu.
- *Unzoom* – see Zoom menu.

10.2.19 Vector Field 3D animated vector view

Description

The animated vector view of the 3D vector field provides a vector plot of the field for each time step in the series. It has much in common with the vector plot for a 2D vector field.

This is the default view for the 3D vector field.

There are four toolbars at the top of the view. These are:

Animation toolbar

- *Current time* – this combo box displays the current value of time. The user can select any value in this box to move to a different time.
- *Go to next time* – steps the display forward one frame.
- *Go to previous time*– steps the display back one frame.
- *Play*– starts the animation forward in time.
- *Play in reverse* – starts the animation backward in time.
- *Pause* – pauses the animation.
- *Go to start* – resets the display to the first frame.
- *Go to end* – resets the display to the last frame.
- *Animation timing delay (ms)* – this text box allows the user to change the speed of the animation. The displayed timing delay is the minimum time between frames. The actual delay may be longer if the drawing overhead is high.

Display toolbar

- *Vectors* – if this is selected the field vectors are displayed. The colours of the vectors can be changed under the *Colour* menu.
- *Grid* – if this check box is selected the grid on which the field is defined is drawn on the graph. Their default colour is white. This can be changed in the Colour menu.
- *Fill view* – if this check box is selected the aspect ratio of the grid is ignored and the graph is expanded to fill the view.

Cursor toolbar

- *x* – displays the x value of the cursor as it moves over the graph.

- *y* – displays the y value of the cursor as it moves over the graph.
- *u* – displays the x component of the velocity field at the location of the cursor as it moves over the graph.
- *v* – displays the y component of the velocity field at the location of the cursor as it moves over the graph.

Scale toolbar

- *Min colour value* – the field value corresponding to the colour at the bottom of the legend. Initially this is set to be the magnitude of the shortest vector on the grid. However, the user may choose to change this value. If vectors shorter than the value specified exist, then these will be coloured white.
- *Max colour value* – the field value corresponding to the colour at the top of the legend. Initially this is set to be the magnitude of the longest vector on the grid. However, the user may choose to change this value. If vectors longer than the value specified exist, then these will be coloured white.

Access

The animated vector view of a 3D vector field can be accessed by double clicking the field in the field calculator manager view or through the popup menu of the field object.

Menus

File menu

- *Save as image…* – enables the user to save the figure as an image file in JPEG, PNG or BMP format. The user can select the resolution of the image.
- *Save to image sequence…* – enables the user to save each figure in the animation as an image in an image sequence. Image file types supported are JPEG, PNG and BMP format. The user can select the resolution of the images. The name, specified by the user, has the number of the frame added as a suffix. It is recommended that the base name ends in an underscore (_) in order to separate the file name from the frame numbering.

View menu

- *Show vectors* – when selected the vectors are displayed. The colour of the vectors can be changed under the *Colour* menu.
- *Show grid* – when selected the grid is displayed.
- *Fill view* – when selected the graph is expanded to fill the entire view.

- *Set vector scale factor…* – allows the user to scale the vectors as these vectors are initially drawn using an arbitrary scale.
- *Set vectors to display…* – by default all vectors defined on the grid are shown. For dense grids this can lead to a congested view. This option allows the user to exclude vectors from the display.
- *Clip vectors* – when selected, any vectors lying outside the displayed region will be clipped.
- *Change field decimal places…*– allows the user to select the number of decimal places used to display the field value.
- *Change position/time decimal places…*– allows the user to select the number of decimal places used to display the independent variable.
- *Hide toolbars*– when selected the toolbars are hidden.

Colour menu

- *Change vector colour…* – allows the user to change the colour of the vectors. The colours corresponding to the maximum, ¾, ½, ¼ and minimum vector magnitudes can be set in a dialog box. The colours between these are interpolated. Buttons allow the default colours and a grey scale range to be created automatically.
- *Change grid colour…* – allows the user to change the colour of the grid lines.
- *Change axes colour…* – allows the user to change the colour of the axes.
- *Change label colour…* – allows the user to change the colour of the labels.
- *Change background colour…* – allows the user to change the background colour.
- *Change adornment colour…* – allows the user to change the colour of adornments drawn on the view.

Animation menu

- *Go to next time* – steps the display forward one frame. This has the same effect as the equivalent button in the animation toolbar.
- *Go to previous time*– steps the display back one frame. This has the same effect as the equivalent button in the animation toolbar.
- *Play* – starts the animation forward in time. This has the same effect as the equivalent button in the animation toolbar.
- *Play in reverse* – starts the animation backward in time. This has the same effect as the equivalent button in the animation toolbar.
- *Pause* – pauses the animation. This has the same effect as the equivalent button in the animation toolbar.

- *Go to start* – resets the display to the first frame. This has the same effect as the equivalent button in the animation toolbar.
- *Go to end* – resets the display to the last frame. The user is prompted to confirm this option. This has the same effect as the equivalent button in the animation toolbar.

Adornments menu

- *Clear adornments* – clears all adornments drawn on the graph.

Zoom menu

- *Zoom* – if a zoom rectangle has been defined (drawn on the view) this command will cause this region to fill the entire display. Data outside this rectangle will be clipped.
- *Unzoom* – reverses any changes to the limits of the display region and restores the full field domain.
- *Clear zoom rectangle* – clears the zoom rectangle.

Popup menu

- *Go to next time* – see Animation menu.
- *Go to previous time*– see Animation menu.
- *Play*– see Animation menu.
- *Play in reverse* – see Animation menu.
- *Pause* – see Animation menu.
- *Go to start* – see Animation menu.
- *Go to end* – see Animation menu.
- *Show vectors* – see View menu.
- *Show grid* – see View menu.
- *Fill view* – see View menu.
- *Zoom* – see Zoom menu.
- *Unzoom* – see Zoom menu.
- Clear zoom rectangle – see Zoom menu.

10.2.20 Vector Field 3D particle track view

Description

The particle track view of the 3D vector field is primarily suited to vector fields corresponding to fluid velocity fields and is very similar to the particle track view of a 2D vector field. The key difference is

that the vector field is now time dependent, with a specified time step and number of frames. The particle motion is now determined by the time varying velocity field.

There are three toolbars at the top of the view. These are:

Animation toolbar

- *Current time* – this combo box displays the current value of time. The user can select any value in this box to move to a different time.
- *Go to next time* – steps the display forward one frame.
- *Go to previous time*– steps the display back one frame.
- *Play*– starts the animation forward in time.
- *Play in reverse* – starts the animation backward in time.
- *Pause* – pauses the animation.
- *Go to start* – resets the display to the first frame.
- *Go to end* – resets the display to the last frame.
- *Animation timing delay (ms)* – this text box allows the user to change the speed of the animation. The displayed timing delay is the minimum time between frames. The actual delay may be longer if the drawing overhead is high.

Display toolbar

- *Particles* – if this is selected the seeded particles are displayed. The colour of the particles can be changed under the *Colour* menu.
- *Vectors* – if this is selected the field vectors are displayed. The colours of the vectors can be changed under the *Colour* menu.
- *Grid* – if this check box is selected the grid on which the field is defined is drawn on the graph. Their default colour is white. This can be changed in the Colour menu.
- *Fill view* – if this check box is selected the aspect ratio of the grid is ignored and the graph is expanded to fill the view.
- *Dimmer* – this slide control allows the user dim or brighten the vector field which acts as a backdrop to this view. When the vector field is displayed at full brightness it can be difficult to clearly see the motion of the particles, and hence dimming the field can be useful.

Cursor toolbar

- *x* – displays the x value of the cursor as it moves over the graph.
- *y* – displays the y value of the cursor as it moves over the graph.

- *u* – displays the x component of the velocity field at the location of the cursor as it moves over the graph.
- *v* – displays the y component of the velocity field at the location of the cursor as it moves over the graph.
- *Drawing tool* – this text field displays the current drawing tool. The drawing tool is set in the *Region* menu.

Access

The particle track view of a 3D vector field can be accessed through the popup menu of the field object.

Menus

File menu

- *Save as image…* – enables the user to save the current figure as an image file in JPEG, PNG or BMP format. The user can select the resolution of the image.
- *Save to image sequence…* – enables the user to save each figure in the animation as an image in an image sequence. Image file types supported are JPEG, PNG and BMP format. The user can select the resolution of the images. The name, specified by the user, has the number of the frame added as a suffix. It is recommended that the base name ends in an underscore (_) in order to separate the file name from the frame numbering.
- *Save tracks to text file…* – enables the user to save the particle tracks to a comma delimited text file (.TXT or .CSV). The x coordinates of track of the first particle are stored first, followed by the y coordinates of the first particle, then the x coordinates of the second and so on.

View menu

- *Show particles* – when selected the particles are displayed. The colour of the particles can be changed under the *Colour* menu.
- *Show vectors* – when selected the vectors are displayed. The colour of the vectors can be changed under the *Colour* menu.
- *Show grid* – when selected the grid is displayed.
- *Fill view* – when selected the graph is expanded to fill the entire view.
- *Set vector scale factor…* – allows the user to scale the vectors as these vectors are initially drawn using an arbitrary scale.
- *Set vectors to display…* – by default all vectors defined on the grid are shown. For dense grids this can lead to a congested view. This option allows the user to exclude vectors from the display.

- *Clip vectors* – when selected, any vectors lying outside the displayed region will be clipped.
- *Change field decimal places…* – allows the user to select the number of decimal places used to display the field value.
- *Change position/time decimal places…* – allows the user to select the number of decimal places used to display the independent variable.
- *Hide toolbars*– when selected the toolbars are hidden.

Colour menu

- *Change vector colour…* – allows the user to change the colour of the vectors. The colours corresponding to the maximum, ¾, ½, ¼ and minimum vector magnitudes can be set in a dialog box. The colours between these are interpolated. Buttons allow the default colours and a grey scale range to be created automatically.
- *Change grid colour…* – allows the user to change the colour of the grid lines.
- *Change axes colour…* – allows the user to change the colour of the axes.
- *Change label colour…* – allows the user to change the colour of the labels.
- *Change background colour…* – allows the user to change the background colour.

Animation menu

- *Go to next time* – steps the display forward one frame. This has the same effect as the equivalent button in the animation toolbar.
- *Go to previous time*– steps the display back one frame. This has the same effect as the equivalent button in the animation toolbar.
- *Play* – starts the animation forward in time. This has the same effect as the equivalent button in the animation toolbar.
- *Play in reverse* – starts the animation backward in time. This has the same effect as the equivalent button in the animation toolbar.
- *Pause* – pauses the animation. This has the same effect as the equivalent button in the animation toolbar.
- *Go to start* – resets the display to the first frame. This has the same effect as the equivalent button in the animation toolbar.
- *Go to end* – resets the display to the last frame. The user is prompted to confirm this option. This has the same effect as the equivalent button in the animation toolbar.

Seeder Menu

- *Display particle seeders…* – displays all of the currently defined particle seeders in a list box. This list box allows the user to create, load and save particle seeders.
- *Load particle seeder…*– loads a previously saved particle seeder from disk.
- *Create particle seeder…*– creates a new particle seeder. The user is given a dialog box in which to enter the particle seeder details (see chapter 16).
- *Generate tracks* – all current particle tracks are deleted and new particle tracks are created. When this item is selected the system creates all of the particle tracks throughout the throughout the time period of the simulation.
- *Delete tracks…* – deletes all particle tracks. The user is prompted to confirm this option.

Region Menu

- *Display regions…* – displays all of the currently defined regions in a list box.
- *Load regions…*– previously stored regions are loaded from a disk file and added to the currently defined regions. Regions are stored in a file with a *.rgn* extension.
- *Save regions as…*– saves all of the currently defined regions in a *.rgn* disk file.
- *Delete all regions* – deletes all currently defined regions. The user is prompted to confirm this option.
- *Delete selected regions* – deletes all selected regions.
- *Select all regions* – selects all regions.
- *Select <-> unselect* – makes all selected regions unselected and vice versa.
- *Invert selected regions* – converts all selected regions that are interior regions into exterior regions and vice versa.
- *Invert all regions* – converts all regions that are interior regions into exterior regions and vice versa.
- *Draw point* – sets the current drawing tool to a point.
- *Draw line* – sets the current drawing tool to a line.
- *Draw internal rectangle* – sets the current drawing tool to an internal rectangle. This is the default.
- *Draw internal circle* – sets the current drawing tool to an internal circle.

- *Draw internal polygon* – sets the current drawing tool to an internal polygon.
- *Draw internal freehand* – sets the current drawing tool to an internal freehand.
- *Draw external rectangle* – sets the current drawing tool to an external rectangle.
- *Draw external circle* – sets the current drawing tool to an external circle.
- *Draw external polygon* – sets the current drawing tool to an external polygon.
- *Draw external freehand* – sets the current drawing tool to an external freehand.

Zoom Menu

- *Zoom…* – causes the view to be zoomed. If only one internal rectangle is currently defined then it is used as the zoom rectangle. If more than one internal rectangle exists then a dialog box is presented to the user, listing all internal rectangles currently defined. The user can select a zoom rectangle from the list.
- *Unzoom*…– this undoes any zoom operation by redisplaying the entire field.

Popup menu

- *Go to next time* – see Animation menu.
- *Go to previous time*– see Animation menu.
- *Play*– see Animation menu.
- *Play in reverse* – see Animation menu.
- *Pause* – see Animation menu.
- *Go to start* – see Animation menu.
- *Go to end* – see Animation menu.
- *Show particles* – see View menu.
- *Show vectors* – see View menu.
- *Show grid* – see View menu.
- *Zoom* – see Zoom menu.
- *Unzoom* – see Zoom menu.

10.3 Popup menus

Each field type has its own popup menu.

10.3.1 Scalar field 0D

The scalar field 0D provides the following options in its popup menu.

- *Save as…* – saves the field to a disk file. The file has a *.s0d* extension.
- *Save as text file…* – saves the field in a comma (,)delimited text file. Either a *.txt* or *.csv* extension can be specified.
- *Open table view…* – opens the table view.

10.3.2 Scalar field 1D

The scalar field 1D provides the following options in its popup menu.

- *Save as…* – saves the field to a disk file. The file has a *.s1d* extension.
- *Save as text file…* – saves the field in a comma (,) delimited text file. Either a *.txt* or *.csv* extension can be specified. The user can choose the save the data with headers (which means header information is included) or without headers (just raw field data is stored).
- *Open graph view…* – opens the graph view.
- *Open PDF view…* – opens the PDF view.
- *Open table view…* – opens the table view.

10.3.3 Scalar field 2D

The scalar field 2D provides the following options in its popup menu.

- *Save as…* – saves the field to a disk file. The file has a *.s2d* extension.
- *Save as text file…* – saves the field in a comma delimited text file. Either a *.txt* or *.csv* extension can be specified. The user can choose the save the data with headers (which means header information is included) or without headers (just raw field data is stored). They can also choose whether they wish to have data stored so that the y axis is downwards or the x axis is downwards (swap axes). This latter

feature can be important when trying to read a field file into a spreadsheet that has a limit on the number of columns available.

- *Open contour view…* – opens the contour view.
- *Open animated graph view…* – opens the animated graph view.
- *Open table view…* – opens the table view.

10.3.4 Scalar field 3D

The scalar field 3D provides the following options in its popup menu.

- *Save as…* – saves the field to a disk file. The file has a *.s3d* extension.
- *Save as text file…* – saves the field in a comma delimited text file. Either a *.txt* or *.csv* extension can be specified. The user can choose the save the data with headers (which means header information is included) or without headers (just raw field data is stored). They can also choose whether they wish to have data stored so that the y axis is downwards or the x axis is downwards (swap axes). This latter feature can be important when trying to read a field file into a spreadsheet that has a limit on the number of columns available. The 2D fields are stored sequentially in the file.
- *Open animated contour view…* – opens the animated contour view.

10.3.5 Vector field 0D

The vector field 0D provides the following options in its popup menu.

- *Save as…* – saves the field to a disk file. The file has a *.v0d* extension.
- *Save as text file…* – saves the field in a comma delimited text file. Either a *.txt* or *.csv* extension can be specified.
- *Open table view…* – opens the table view.

10.3.6 Vector field 1D

The vector field 1D provides the following options in its popup menu.

- *Save as…* – saves the field to a disk file. The file has a *.v1d* extension.
- *Save as text file…* – saves the field in a comma delimited text file. Either a *.txt* or *.csv* extension can be specified. The user can choose the save the data with headers (which means header information is included) or without headers (just raw field data is stored).
- *Open graph view…* – opens the graph view.

- *Open vector view…* – opens the vector view.
- *Open table view…* – opens the table view.

10.3.7 Vector field 2D

The vector field 2D provides the following options in its popup menu.

- *Save as…* – saves the field to a disk file. The file has a *.v2d* extension.
- *Save as text file…* – saves the field in a comma delimited text file. Either a *.txt* or *.csv* extension can be specified. The user can choose the save the data with headers (which means header information is included) or without headers (just raw field data is stored). They can also choose whether they wish to have data stored so that the y axis is downwards or the x axis is downwards (swap axes). This latter feature can be important when trying to read a field file into a spreadsheet that has a limit on the number of columns available.
- *Open animated graph view…* – opens the animated graph view.
- *Open vector view…* – opens the vector view.
- *Open particle track view…* – opens the particle track view.
- *Open table view…* – opens the table view.

10.3.8 Vector field 3D

The vector field 3D provides the following options in its popup menu.

- *Save as…* – saves the field to a disk file. The file has a *.v3d* extension.
- *Save as text file…* – saves the field in a comma delimited text file. Either a *.txt* or *.csv* extension can be specified. The user can choose the save the data with headers (which means header information is included) or without headers (just raw field data is stored). They can also choose whether they wish to have data stored so that the y axis is downwards or the x axis is downwards (swap axes). This latter feature can be important when trying to read a field file into a spreadsheet that has a limit on the number of columns available. The 2D fields are stored sequentially in the file.
- *Open animated vector view…* – opens the animated vector view.
- *Open particle track view…* – opens the particle track view.

11 Particle seeders

11.1 Description

Particle seeders are support objects employed in the 2D vector field and 3D vector field particle tracking views (see sections 10.2.17 and 10.2.19).

A particle seeder has a simple job to perform. It is to introduce particles into a vector field (interpreted as a velocity field by the particle seeder) and then produce tracks for these particles by determining the position of each particle in the flow at each time step, using the defined velocity field. If a particle leaves the flow domain, or encounters a region in which the velocity is undefined, during its motion, then that particle track terminates at that point.

Particle seeders employ used defined regions (drawn in one of the particle tracking views) to specify the locations in the flow at which the particles are to be introduced. These regions can be single points, lines, or any of the other internal/external regions supported by the *ImageStream* system. Particles can be introduced at one time and tracked from that point, or they may be introduced at a number of times. The particles can be distributed uniformly or randomly in the regions in which they appear and the user has a number of choices on how the velocity field at each particle's location is to be evaluated.

Particle seeders can be saved to disk in files with a .pts extension and reloaded at a later time.

11.2 Views

The particle seeder object has a details view.

11.2.1 Details view

Description

The details view displays the parameter settings for the particle seeder object. Three tabs are used to organise the data. These tabs,

and the information they display, match those described in the section on the creation of particle seeder objects (see section 16.4).

Note that particle seeder objects cannot be changed through their details view.

Access

Each field view that utilizes particle seeders provides a view listing all of the currently defined particle seeders. The particle seeder details view can be accessed by double clicking a particle seeder object displayed in this list box, or through its popup menu.

11.3 Popup menu

The PTV analysis provides the following options in its popup menu.

- *Save as…* – saves the particle seeder object in a disk file. All particle seeder files have the extension *.pts*.
- *Change name…* – this option allows the user to change the name of the seeder.
- *Edit parameters…* – this option opens a dialog box that allows the user to change the particle seeder's parameters.
- *Open details view* – opens the object's details view.
- *Close all views* – closes all views for this object which are currently open.

11.4 Creation

A particle seeder object can be created in the particle tracking view, either through a menu bar option, or through the view displaying the list of currently defined particle seeders. See sections 10.2.17 and 10.2.19 for details.

A dialog box with 3 tabs is displayed. These tabs specify information in 3 general areas, and are discussed in the following sections.

11.4.1 Name

The *Name* tab allows the user to enter a name for this particle seeder that will identify it amongst other particle seeders. The user enters the chosen name into the name text field.

11.4.2 Seeding parameters

These parameters govern the manner in which the particles are seeded into the flow and how their paths are determined.

The following parameters are set here:

- *Start seeding at* – this combo box specifies the first frame (time) in the simulation at which seeding occurs.
- *Finish seeding at* – this combo box specifies the last frame (time) in which seeding occurs.
- *Seed every* – this field specifies how often particles are introduced into the flow between the start and finish frames. If this parameter is set to 1, then particles are introduced at every time step between the start and finish frames.
- *Particles per seeding* – this parameter specifies how many particles are introduced into each seeding region at each seeding. If the user wishes to use different numbers of particles for each region then separate seeders should be created for each region. It is important to recognise that seeding a large number of particles often into a flow with many frames will require substantial memory and processing time, and should be avoided if possible.
- *Particle distribution* – the user can choose to seed the particles randomly of uniformly within the seeding regions.
- *Cell interpolation* – this parameter specifies how the velocity at a particle location is calculated. The grid cell in which the particle resides is identified and a velocity estimate is available provided the velocity at all corners of the cell is defined. The estimated velocity can be obtained by averaging the four corner velocities, or by doing a bi-linear fit to the velocity field and interpolating the velocity at the particle location. The bilinear fit is the default setting.
- *Velocity estimate* – the displacement of the particle can be based on the velocity field at the current time step (forward), the velocity field at the next time step (backward), or a combination of the two (central). The central estimate is the default setting.
- *Particle colour* – the colour used to render the particles generated by this particle seeder is painted in the rectangular box. The colour can be changed by clicking the *Change colour* button below.

11.4.3 Seeding regions

The regions in which the particles are introduced can be selected in this tab. The currently defined regions for the field view that is to be seeded with particles are listed in the top list box. The regions used by the seeder are listed in the bottom list box. Regions can be

selected in the top list box and added to the seeding region list, and regions selected in the seeding region list can be deleted. There is no limit on the number of regions used for seeding.

12 Summary of changes in V6.00

12.1 Description

This chapter provides an overview of the changes (other than minor fixes) that have been implemented in *ImageStream* 6.00.

1. Each user of *ImageStream* 6.00 must register the application using a registration key provided by the software's author, Dr Roger Nokes.

2. The variety of different user interface looks and feels are now available. Section 2.2.1.

3. The convert format transformation is new (see chapter 4).

4. The convert type filter has been added to the list of available filters (see chapter 6).

5. The size of images are checked when an image sequence is created to ensure consistency in future processing.

6. Eight new field views have been introduced. These are:

- Scalar Field 1D PDF view
- Scalar Field D spectral view
- Scalar Field 2D animated graph view
- Scalar Field 2D multi graph view
- Vector Field 1D graph view
- Vector Field 2D animated graph view
- Vector field 2D particle track view
- Vector Field 3D particle track view

Information on all of these views can be found in chapter 10.

6. The protocols for drawing regions on the screen, accessing popup menus and so on have been changed. See chapter 5 for details.

7. Significant changes have been made to other field views, particularly the Vector Field 1D vector view. All graphical views

now provide a zoom capability, the option to hide the toolbars, and provide a popup menu for easy access to certain view functionality.

8. The interfaces and functionality of all views have been modified to ensure a consistency between them.

9. The two dimensional table views have been modified so that the row labels no longer scroll with the data columns.