Getting Started with DirectAid 2.0

Starting DirectAid

You may get a message on startup indicating that your graphics card does not support some features used in DirectAid. The abilities of modern graphics cards go well beyond simply drawing pixels to the screen. Rendering which was once carried out by the CPU can now be done on a specialized graphics processing unit onboard the graphics card. In some cases, it may be necessary to update your graphics drivers before DirectAid will work smoothly. If you are using a desktop computer, these drivers can be obtained from your graphics card manufacturer. If you are using a laptop computer, you will likely need to obtain updated drivers from the laptop manufacturer. If you are having problems locating suitable drivers, feel free to contact us and we will try to help.

The Application Window

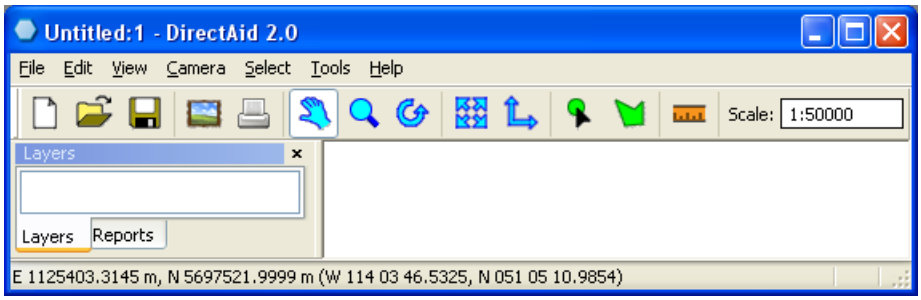


Figure 1 - The application window

When you first open DirectAid, you should see a screen similar to the one shown in Figure 1. We will first describe the basic parts of the application window. At the very top is the *title bar*. This contains the application title, system menu, and buttons for minimizing, maximizing, and closing the program. Just below this is the *menu bar*. There are seven menus, whose commands will be described in greater detail later on:

- The *File* menu contains commands for opening and saving DirectAid workspaces, importing data into a workspace, printing, and closing the program.
- The *Edit* menu contains commands for cutting, copying, and pasting items, as well as commands for undoing and redoing actions.
- The *View* menu contains commands for showing and hiding the different components of the interface, as well as the *Preferences* command for setting program options.
- The *Camera* menu contains commands for changing the camera.
- The *Select* menu contains commands for setting the selection mode.
- The *Tools* menu contains commands to activate the *Rechain*, I*nterpolate*, and *View Templates* tools.
- Finally, the *Help* menu contains a link to this document, *Getting Started*.

Below the menu bar is the *toolbar*. You will likely be familiar with some of the buttons, and not with others. You can quickly find out what a particular button does by holding

the mouse cursor over the button for a moment. A small description will appear just below the button, and a slightly longer description may appear in the status bar below. The buttons on the toolbar are as follows:

Create a new workspace
Open an existing workspace
Save the active workspace
Set the print area
Print the active map view
Pan the camera
Zoom the camera
Rotate the camera
Un-zoom the camera
Square camera axes
Select map objects
Select objects in area
Measure length and area

Below the toolbar, on the left, is the *layer view*. Here you will see a list of all the layers which have been loaded into the current workspace. To the right is the *map view*.

Finally, at the bottom of the application window is the *status bar*. The left portion of the status bar displays context-sensitive information. If the mouse is over a toolbar button, for example, this will show a description of the button. If the mouse is over the map view, the status bar will show the position of the mouse cursor.

The Layer View

By default, the layer view is located to the left of the map view. By dragging its title bar, the layer view can be moved to any position on the screen, or docked on an edge of the application window. To show or hide the layer view, choose *Layer Bar* in the *View* menu.

The layer view lists imported layers in a tree structure. Layers are drawn from bottom to top, so that a layer at the top of the layer view is drawn on top of all other layers. A layer can be expanded by clicking the ⊞ symbol to the left of the label, or collapsed by clicking the ⊟ symbol. A layer can also be expanded or collapsed by double-clicking the label.

To show or hide a layer, click the check box to the left of the label. If a parent layer is hidden, its children will not be drawn.

Some layers, such as folders, can be renamed. To rename a layer, either click on the label of an already-selected layer, or hit *F2* with the layer selected. This will enter label edit mode.

To select a range of layers, click on the first layer, then hold the *Shift* key while clicking on last layer. To add to an existing selection, hold the *Ctrl* key while selecting. To add a range to an existing selection, hold the *Ctrl* key and click on the first layer of the range, then hold both the *Shift* and *Ctrl* keys and click on the last layer.

Some layers can be dragged from one place to another, for example to change the drawing order or to arrange layers into folders. The location where a dropped item will be placed is indicated either by a selection, in which case the item will be inserted as a child of the selected item; or as a line between two items, in which case the item will be inserted between the two items. If no selection or line appears, this indicates that the item cannot be dropped at the current location.

By right-clicking on a layer, you can bring up a context menu with common operations for the layer, usually including *Delete* and *Properties*. By right-clicking on the white background in the layer view, it is possible to add items at the root level.

If the camera is in Earth Centered Earth Fixed (ECEF) coordinates, you will see an additional layer called *Globe Layer*. The globe can be drawn either as a simple grid or using NASA's Blue Marble image. To change the globe mode, right-click on the *Globe Layer* and select a mode from the context menu.

Layer Properties

By right-clicking on certain layers, you can access the *Properties* command. Properties for top-level layers usually include the three standard coordinate system pages. In addition, properties may include symbol, line, and text colours and formats, as well as a page with general information about the layer. To accept changes made in the property sheet, click the *OK* button at the bottom. To close the sheet without applying changes, click *Cancel*.

Layer properties can be used to change an incorrectly assigned coordinate system tag. It is important to remember that changing the coordinate system for a layer does not change the underlying data, but rather changes the meaning of that data. For example, if we had imported a project but mistakenly assigned it the NAD83 datum, we could later change this to NAD27 in the property sheet, and the effect would be the same as if we had given it the correct datum in the first place.

In some cases, a parent layer will contain property sheets relating to its children. In these cases, the parent property sheet is populated with the options you might see in the child property sheets. If a particular option is set the same in all children, that value will be selected in the parent property sheet. Otherwise, the option will be present but no value

will be selected. Selecting a value in the parent property sheet assigns that value to all of the children.

The Map View

Panning, Zooming, and Rotating

You can switch the mouse function between pan, zoom, and rotate modes either using the buttons on the toolbar or by choosing the appropriate option in the *Camera* menu.

To pan the map while in pan mode (), click and drag with the left mouse button. You will see the map move with your cursor. If you have a middle button or wheel on your mouse, you can pan in any mode by pressing the middle button (or wheel). To use this function, click the middle button and then move the mouse away from the original position. A small crosshair will appear at the position where you pressed the button. The page will move in a direction toward the cursor with a speed which is proportional to your distance from the stationary crosshair. If you click-and-drag the middle button, the pan will end when you let the button up. Otherwise, the pan will end when you click the middle button a second time.

To zoom the map while in zoom mode (), click and drag with the left mouse button on the map to define a zoom box. When you let the button up, the camera will zoom to fit that region to the page. To zoom out while in zoom mode, just click the left mouse button without moving it. If you have a mouse wheel on your mouse, you can zoom in and out in any mode using the wheel. You can control the zoom step, as well as the direction (whether pulling the wheel toward you zooms in or out) using the *Preferences* command in the *View* menu.

To rotate the map, you must be in rotate mode (). When you enter this mode, you will see a small crosshair appear on the screen. This is the point about which the camera will be rotated. To move the point, just click the left mouse button anywhere on the page. To rotate around that point, click and drag with the left mouse button.

To un-zoom the map, click the *un-zoom* button (). If you have loaded project layers, this will un-zoom to the extent of the projects. If you do not have projects visible, it will un-zoom to the entire extent of the workspace. If you wish to square your camera axes, so that right is east and up is north, click the *square axes* button ().

Coordinate Systems

In DirectAid, each piece of imported data is tagged with a *source coordinate system*, including the following information:

- Units of measurement
- Geodetic datum
- Projection

Every time you import something, you will be prompted for this information. Similarly, the map view is associated with an independent coordinate system. This is defined in the *Camera/Options* dialog box.

The camera coordinate system

Each layer in DirectAid is stored in its source coordinate system, along with a tag indicating what that source coordinate system is. The actual conversion to camera coordinates occurs on-the-fly.

To change the camera coordinate system, choose *Options* under the *Camera* menu. The options are broken into three groups. The first group, *Units*, is shown in Figure 2. This page controls the units used when displaying positions, lengths, and areas throughout DirectAid. To change units, just click the drop-down to the right of the unit.

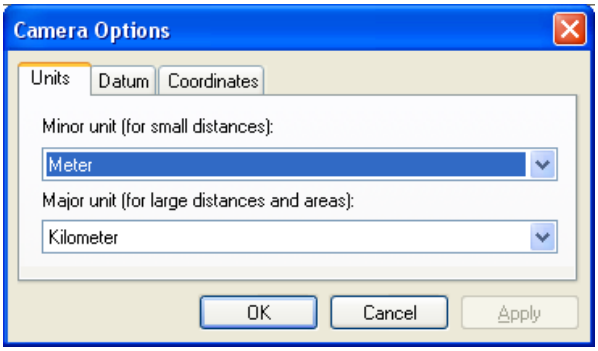


Figure 2 - Camera units

The first time you do this, you will see a full unit list, as shown in Figure 3. In subsequent uses, you will see an abbreviated list, as shown in Figure 4. To view the full list, just click on “more options”.

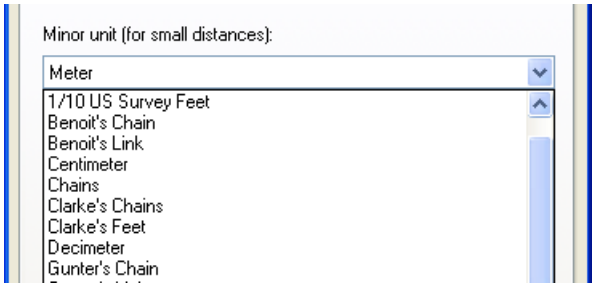


Figure 3 - Full unit list

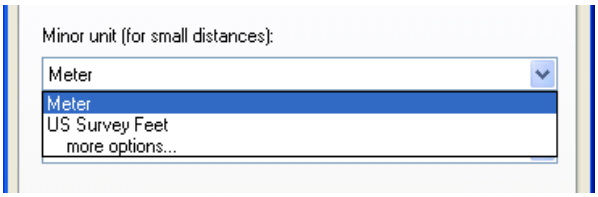


Figure 4 - Abbreviated unit list

The second group of options, *Datum*, is shown in Figure 5. Once again, the first time you set the camera options, you will see a full list of datums. Subsequently, you will see only an abbreviated list.

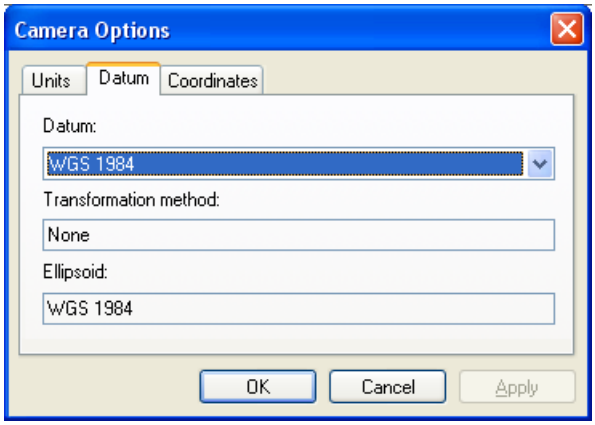


Figure 5 - Camera datum

DirectAid is aware of hundreds of datums, but the most commonly used datums used in Canada are outlined in Table 1.

Table 1 - Common Canadian datums

Datum	Description
NAD 1927 - Canadian Transformation	NAD27 using NTV2
North American Datum 1983	NAD83
WGS 1984	WGS84

Finally, the third group of options, *Coordinates*, is shown in Figure 6. Library projections are divided into "groups". For example, "Universal Transverse Mercator" is one such group. As usual, the group and coordinate system lists come in "full" and "abbreviated" versions. The library projections include thousands of coordinate systems used around the world.

One particular coordinate system deserves special mention. If the group is set to "XYZ Cartesian ECEF" and the coordinate system to "Earth Centered Earth Fixed", the map will be as though the camera were looking down on the earth from space. In this coordinate system, if you zoom out far enough, you should see the globe.

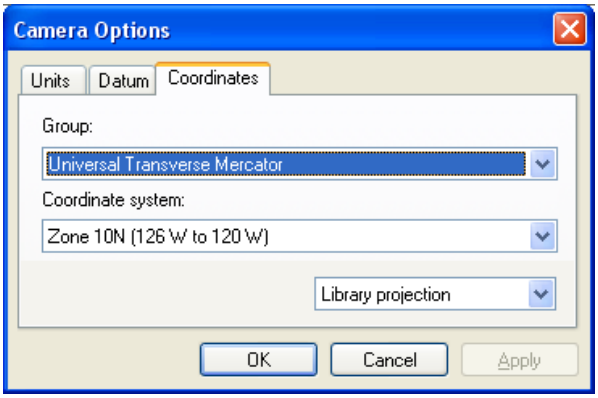


Figure 6 - Camera coordinates

If you are unable to find the projection you're looking for, please contact us and we may be able to tell you where it is located. If, however, your desired projection simply has not been included in the library (or if you are using a parametric projection such as a non-standard Lambert Conformal Conic projection), it is possible to define a custom projection. To do this, click on the drop-down next to "Library projection" in the lower right and change to "Custom projection", as shown in Figure 7.

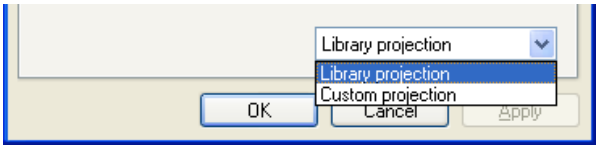


Figure 7 - Changing to custom projection

The Coordinates page will change as shown in Figure 8. You may now choose a general projection and specify the parameters yourself.

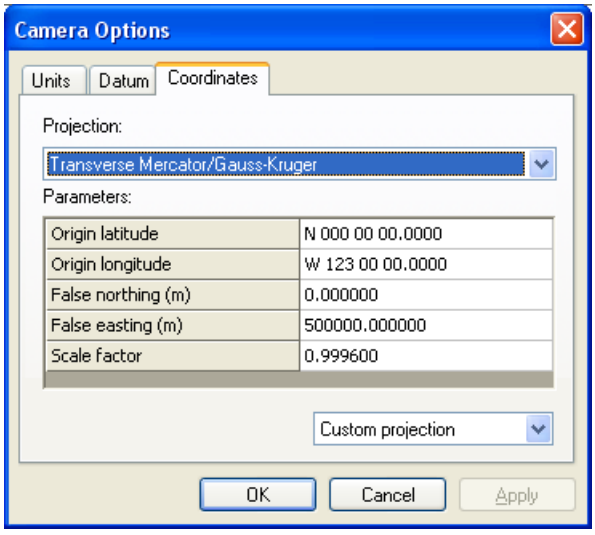


Figure 8 - Custom projections

Once you have made changes to the camera coordinate system, you can click *OK* to apply those changes and close the options dialog box. Your data will automatically be shown in the new coordinate system. Although DirectAid attempts to transform the camera position so that you are looking at the same place in the new coordinate system, you may need to un-zoom ( ) and re-square your camera axes ( ) when you change the camera coordinate system.

Accuracy and error indicators

On the right of the status bar are two indicators. When the indicators are lit up, they will appear as in Figure 9 below. On the left is the accuracy (ACC) indicator. On the right is the error (ERR) indicator.

Figure 9 - Accuracy and error indicators

DirectAid uses a progressive scheme to transform source data on-the-fly. That is, it will transform data in "idle time" while your processor isn't doing anything else, and it will update your display as the data is transformed to a finer resolution. The *accuracy indicator* will light as long as this process is incomplete. You may occasionally notice a background image updating as you zoom in and out, or when you pan to a different part of the survey. This is normal. When the accuracy indicator goes off, that is your guarantee that the data has been transformed to within one pixel error. That is, because your display has an inherent inaccuracy, the transformed data is indistinguishable from the exact transform.

When printing, DirectAid will ensure that the workspace has been refined to the appropriate error threshold before sending data to the printer (or to a file if you are printing to file).

The *error indicator* will light when some part of a transform has failed. This usually indicates that either the source or camera coordinate system is not valid in the region containing your data. For example, the datum "NAD 1927 - Canadian Transformation" is valid only within Canada. You may see the error indicator light when transforming an image which spans the boundary of the datum. In this case, DirectAid will try to determine the region in which the datum is valid, and you will see only the part of the image for which the datum is valid.

Saving and Opening a Workspace

A workspace can be saved as a DirectAid workspace file. These files include all project and vector data contained in the workspace, as well as image georeferencing information. In order to prevent duplication of large amounts of image data, this data is not saved as part of the workspace at this time. However, it will be loaded automatically from the

source image when the workspace is opened. If the source image cannot be located, you will be given the option to browse for the image data.

To save a workspace to disk, choose *Save* or *Save As* from the *File* menu, or use the toolbar button (). You will be prompted for a file name. As usual, DirectAid will add the appropriate extension to the file name if you do not enter one yourself.

DirectAid 2.0 workspace files can be recognized by their icon (). These files can be opened by double-clicking the file in Windows Explorer. Files can also be opened from within DirectAid by choosing *Open* from the *File* menu, or using the toolbar button ().

To create a new workspace, either choose *New* from the *File* menu, or click the button on the toolbar ().

Projects

Building a project

To build a new project within the current workspace, right-click in the white area of the layer view. From the context menu, choose *New* and then *Project layer*. This will bring up the *Project Wizard*, which can be used to build projects of several types. The first page of the wizard is shown in Figure 10.

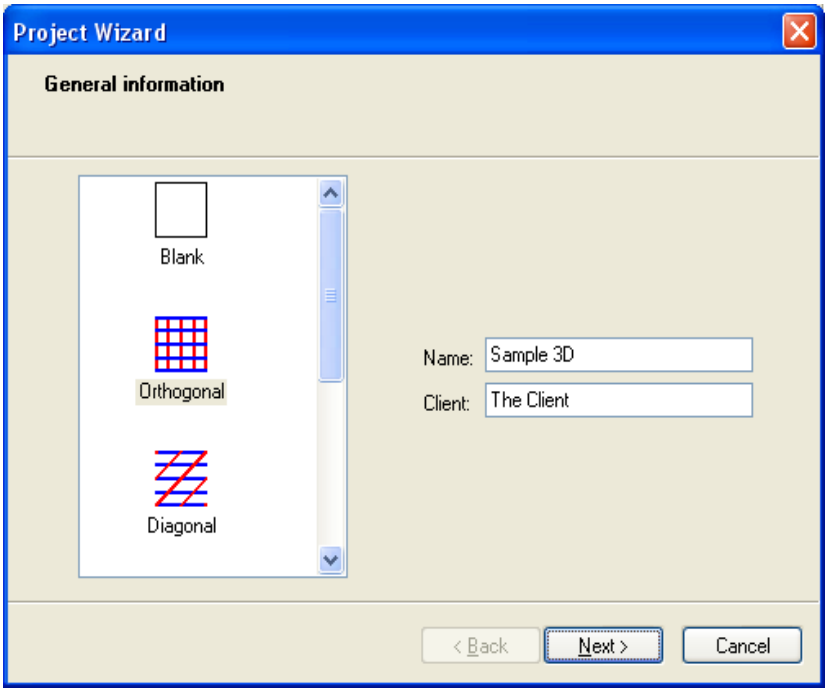


Figure 10 - Project wizard: general information

In the *General information* page, you can select the type of survey you want to build, as well as assigning project name and client labels. To select a project type, click on the icon

in the list on the left. When you have entered this information, click *Next* to continue. This will lead you to the standard three coordinate system pages.

The coordinate system specified on these three pages is used to build the project. The units you specify will be used to define the project position and extent. The datum and coordinate system you specify will be used to define "north" and "east" when building the project.

After you have specified a coordinate system, clicking *Next* will take you to the *Position* page, shown in Figure 11.

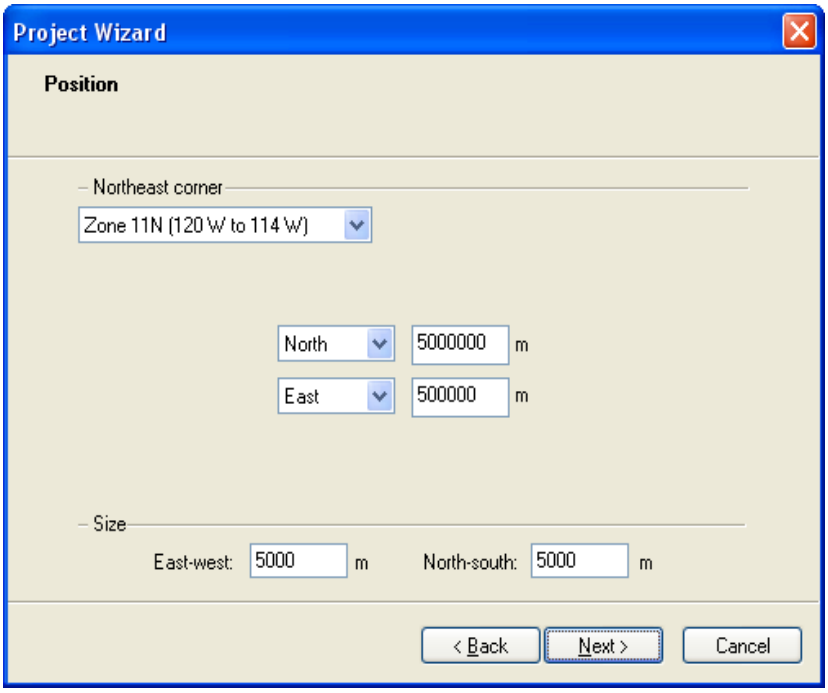


Figure 11 - Project wizard: position

The position of the survey can be specified either in grid coordinates or in geodetic (latitude/longitude) coordinates. To select one of these two methods, click on the drop-down list at the top of the page. When entering the northeast corner position and size of the survey, take note of the units. These are the units you specified in the coordinate system pages. If you would like to change the units to something more convenient, click the *Back* button to return to the *Units* page.

You may also be able to specify the northeast corner of your project in terms of the Dominion Land Survey (DLS) or National Topographic System (NTS) grids if your project is in Canada. To use DLS grids, you will need to purchase a copy of the section corner database from Terra Management (see "DLS grids" on page 26). When you have entered a position and size for the survey, click *Next*.

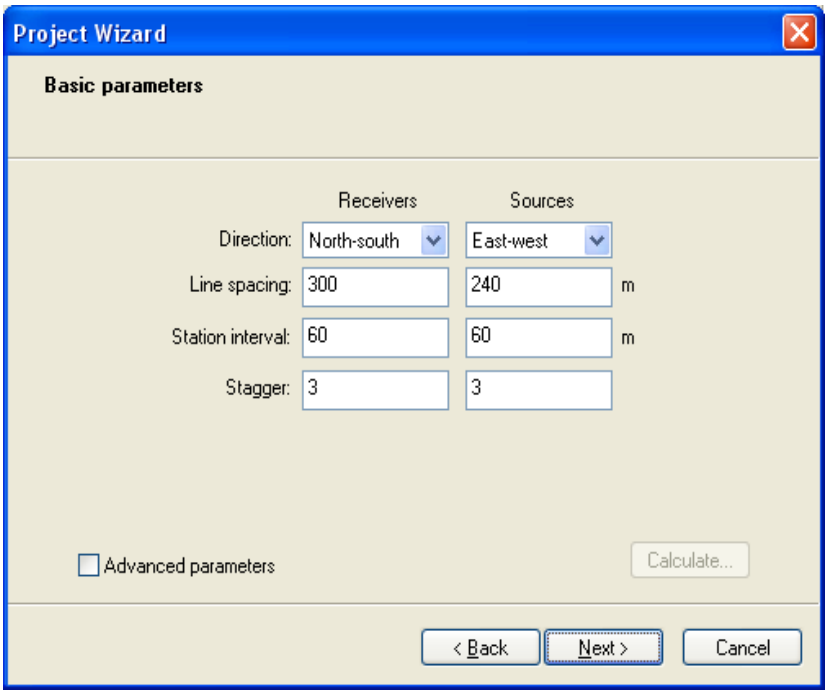


Figure 12 - Project wizard: basic parameters

The *Basic parameters* page, shown in Figure 12, is used to specify the direction of receiver and source lines, line spacing, station interval, and a stagger period.

The stagger period is used to shift lines incrementally in the inline direction. For instance, a value of "3" results in what is known as a "triple stagger" survey. In this case, the stations of every third line would line up. If you have chosen a line spacing which is a multiple of your station interval, a triple stagger would result in a 3x3 grid of midpoints within each bin, whereas no stagger (a stagger value of "1") would result in midpoints centered in the bin.

By clicking the *Advanced parameters* checkbox at the bottom of the page, you can change two additional parameters, shown in Figure 13. These can be used to introduce an offset in line or station position.

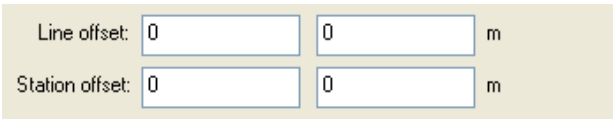


Figure 13 - Project wizard: advanced parameters

Depending on the type of survey you have selected, you may encounter one or more additional *Basic parameters* pages following the first one. For diagonal projects, you be asked to specify either a rotation or slope for the diagonal, as shown in Figure 14. A rotation/slope of "0.0" corresponds with an orthogonal survey.

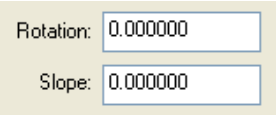


Figure 14 - Diagonal project options

For a brick survey, you will need to specify the brick period, as shown in Figure 15. In a brick survey, the source salvos are offset in the crossline direction by a fraction of the source line spacing. This number of salvos between repeated offsets is called the *brick period*.

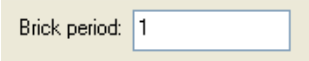


Figure 15 - Brick project options

When you have finished entering basic parameters, click *Next* to move on to the *Line/station numbering* page, shown in Figure 16.

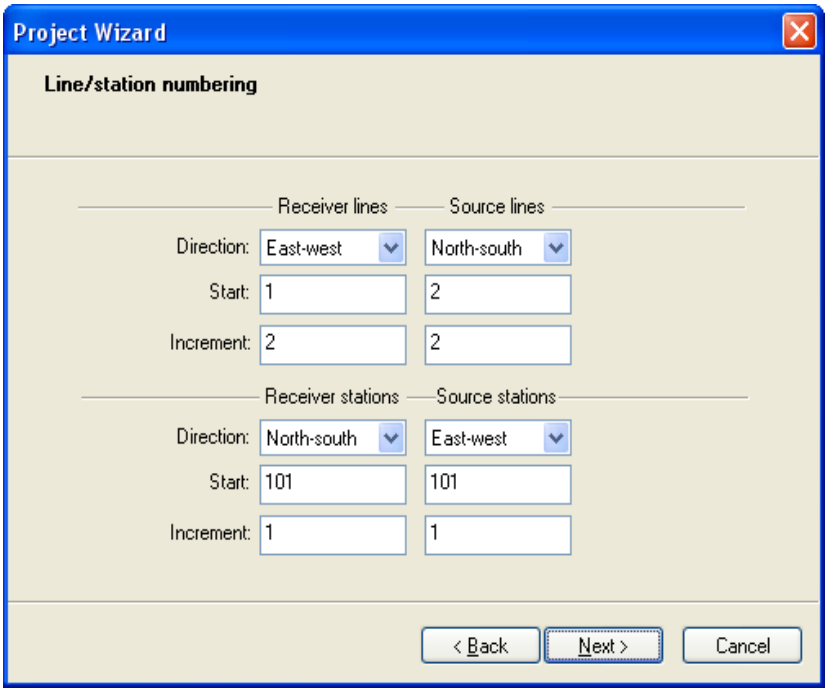


Figure 16 - Project wizard: line and station numbering

You will probably accept the defaults on this page most of the time. However, you have the option here to change which corner your lines and stations are numbered from, what the first line/station number is, and what the increment is. When you have finished, click *Next*. You will be shown a summary of the parameters you have entered. To build the project, click *Finish.* The project is added to the workspace, but the map is not changed. To view the project, you may need to un-zoom ( ) and re-square your camera axes ( ).

Importing and exporting projects

Importing a project

To import a project, choose *Project layer* in the *Import* pop-up menu. First you will be asked to select the files to import. For project layers, all files imported in a single operation must be referenced to the same coordinate system, and must share the same layout. Once they are imported, these files will be grouped under a single project layer. When you have selected one or more files, click *Open* to continue.

Next, the project import wizard will appear. The first page of the wizard is shown in Figure 17. In this page, you may specify the file layout. At the top right of the page is a drop-down box which tells you which file you are currently viewing. Because all the files must share the same layout, you can use this control to switch between files and make sure the columns are the same in each file.

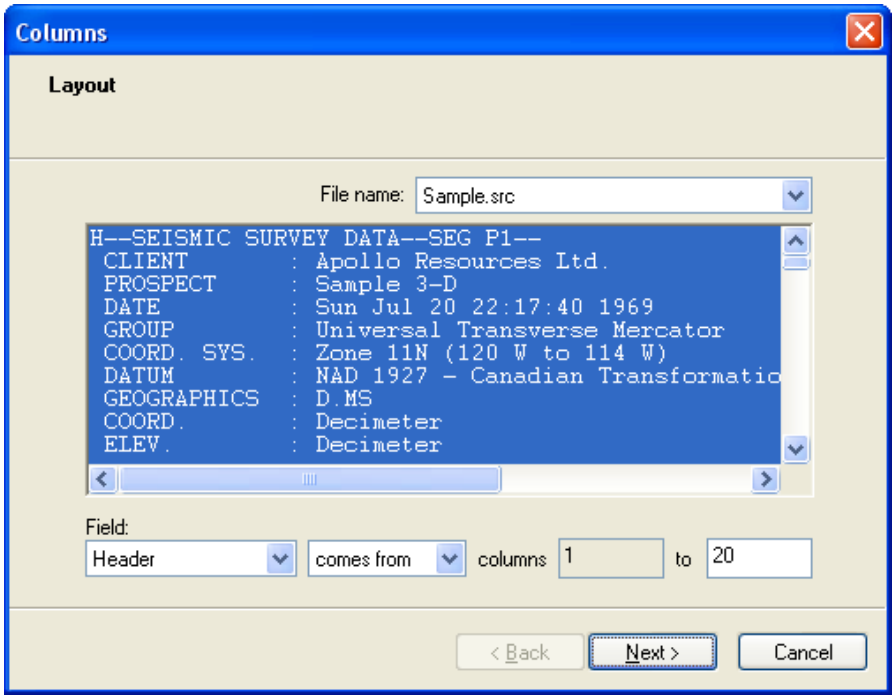


Figure 17 - Project file layout

Below the file selection drop-down is the file preview. Using this control you can scroll through the entire file to make sure it fits the specified layout. In addition, you will notice that a range of rows or columns is selected in the preview. This corresponds with the currently-selected field (indicated in the drop-down box below the preview). You can change the position and width of a range by clicking and dragging the middle or edges of the range using the left mouse button.

Below the preview is an area used to assign ranges of columns in the SEG-P1 file to specified properties in the imported project. By default, the “Field” dropdown is

populated with the properties that are needed to create a valid project: header, type, line, station, easting, northing, and elevation. Most of the time, you will simply want to check that these fields are associated with the correct columns. To do this, select each field in turn and scroll through the preview to ensure that the required information sits inside the highlighted columns.

In some cases, you will want to assign user-defined properties to the stations. By default, user-defined properties are not set when you import a project. However, you may initialize fields using data in the SEG-P1 file (as in Figure 18), or set them to a fixed value (as in Figure 19).

Figure 18 - Custom field initialized from SEG-P1 file

Figure 19 - Custom field initialized to fixed value

To assign values to a field, first select the field from the first drop-down box on the left. Next, choose the initialization method. Finally, enter either a range of columns or a fixed value for the field.

This is a good time to make a note of any information which is available in the header of the files you are importing, such as datum and projection. When you have made a note of this information, click *Next* to continue.

What follows are three wizard pages which are very similar to the camera options pages described in “The camera coordinate system” on page 5. Use these three pages to specify the source coordinate system for the imported files. If you are unsure about this information, you may want to make educated guesses, since you can always change the source coordinate system later on. When you have reached the last wizard page, click *Finish* to import the files.

If the files contain stations with duplicate line/station numbers, the *Resolve Duplicates* wizard, shown in Figure 20, will appear. A single group of duplicates is shown in red, while the surrounding text is shown in gray. The selected line, marked with a green bar, will be kept. The others will be discarded. To move between groups of duplicates, use the *Previous* and *Next* buttons. When you have checked all the duplicates, click *Finish* to continue.

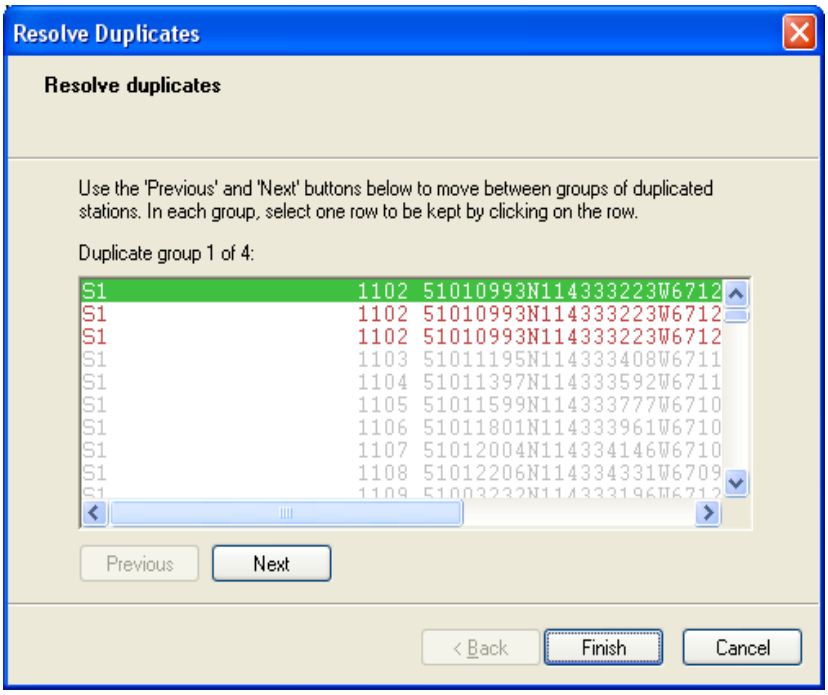


Figure 20 - Resolving duplicates

If the files contain stations with indeterminate type, The *Specify Line Types* wizard, shown in Figure 21, will appear. To move a line from one column to the other, select it and click the appropriate arrow in the middle. When you have assigned all the line types correctly, click *OK* to continue.

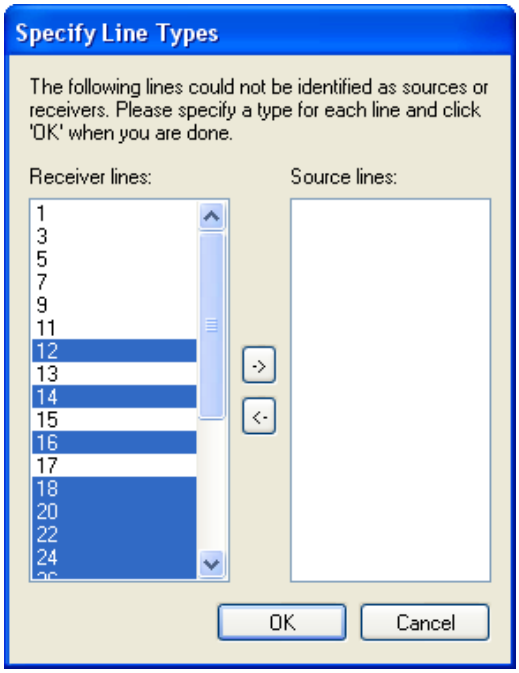


Figure 21 - Specifying line types

When the import operation is complete, DirectAid will create a new layer in the layer view. You may need to un-zoom ( ) in order to see the project. If you still cannot see the project, check if the error indicator is lit. This may indicate that you have chosen a datum or projection which is not valid for the project area.

Exporting a project

A survey layer is the immediate child of a project layer, and represents a group of project lines. Typically, a project layer will have a child survey layer for sources, and another for receivers. A survey layer can be exported to a SEG-P1 file, an SPS point file, or a comma-separated value (CSV) file.

To export a survey layer, right-click on the "Source" or "Receiver" layer in the project to access the context menu, then choose the *Export* command. You will be prompted for a file name and type. DirectAid 2.0 will add the appropriate extension to the file name if you do not enter one yourself. When you have done, click *Save* to continue. Next, you will be presented with the survey export wizard. This wizard consists only of the three coordinate system pages. Default values will be the same as the source coordinate system for the parent project. Your choices here affect only the exported data, and will have no effect on the project layer itself. Files are written in the standard format as outlined by the Society of Exploration Geophysicists (http://seg.org/publications/tech-stand/).

When writing SPS point files, by default the point code (columns 27-28) will be blank. The point code consists of two characters which indicate the station type (e.g. geophone, hydrophone, vibroseis, explosive, air gun). The specific point code used for each station type is not defined by the SPS specification, so we've left this up to the user. To set the point code for a group of stations, first select the stations. Right-click on the selection and choose *Properties* from the context menu. If needed, add the "_SPS_Point_Code" built-in property, as described in "Setting properties" on page 19. Enter a one- or two-character point code in the text box and click *OK* to make the change.

The wizard for exporting to CSV is similar to the SEG-P1 and SPS export wizards, except that it includes an initial page for specifying which data should be written to the file, as shown in Figure 22.

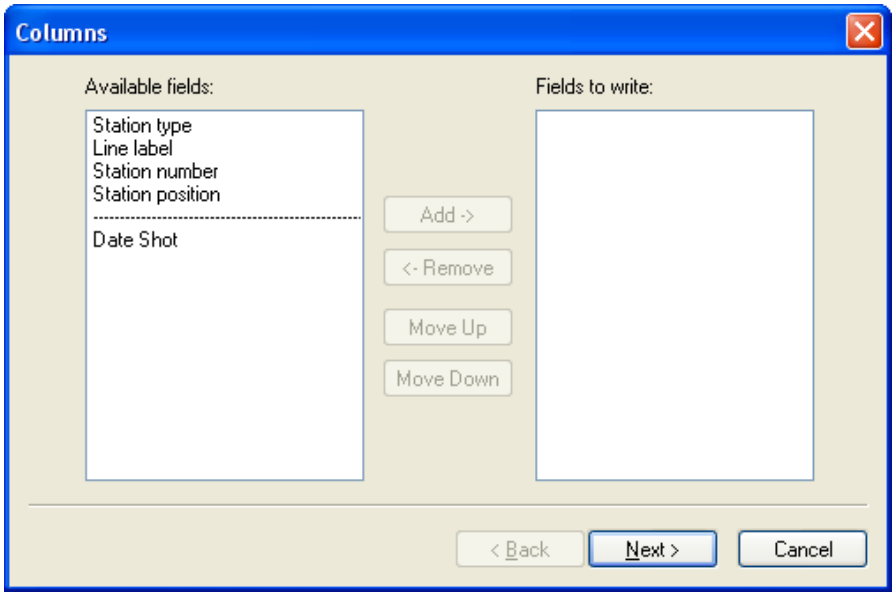


Figure 22 - CSV columns

Fields can be moved between the “Available fields” column and the “Fields to write” column, or re-arranged within the latter. Note that both built-in fields and user-defined fields are available to be written. When you have selected the fields to be written, click *Next* and continue as for SEG-P1 export.

Project properties

Each project has several attributes which can be accessed through the *General* tab of its *Properties* dialog box (see “Layer Properties” on page 3). These attributes are shown in Figure 23.

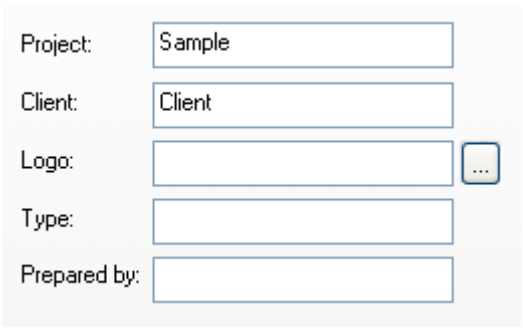


Figure 23 - Project properties

The values in these fields are used to populate the header of exported SEG-P1 and SPS files, and also for labels in some reports.

Flexible Properties

It can sometimes be helpful to be able to assign properties to a layer. These properties can then be displayed in the map, used as a filter in some operations, and inserted into labels and reports.

The basis for DirectAid's property mechanism is the *field*. Fields allow us to define the type of data and how it will be displayed. For example, one possible field would be "hole depth". Hole depth is a number. It might be displayed as a circle around the shot, with a radius which depends on depth.

A *property* is a particular instance of a field. For example, some shots may have a hole depth while others do not. Once the field is defined, we can add a property of that type to any layer. In the example above, a particular shot might have a hold depth of "20.0".

Managing fields

To add new fields, use the *Manage Fields* tool, located in the *Tools* menu. The tool is shown in Figure 24.

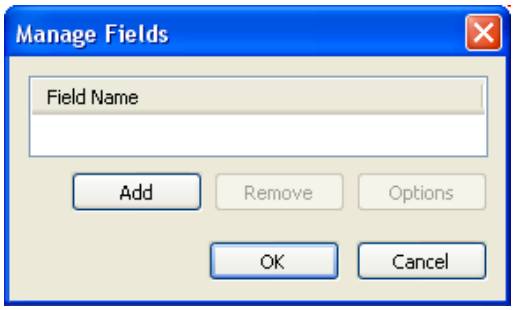


Figure 24 - The Manage Fields tool

To add a new field, click *Add*. This will display the *Field Properties* dialog box, shown in Figure 25.

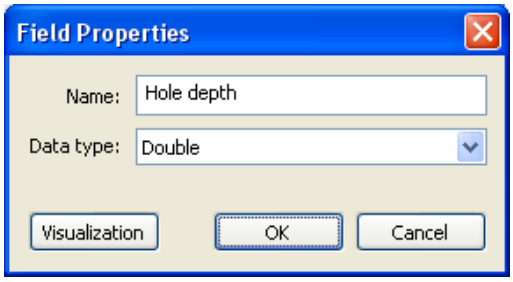


Figure 25 - The Field Properties dialog

First, we need to give a name to the field. This is the name that will be used to identify the field in property dialogs, filters, and reports. Using our example above, a good name

might be “Hole depth”. Finally, we need to choose a data type for the field. These data types are described in the table below:

Data Type	Description
Boolean	True or false.
Double	A real number (e.g. 3.1415).
Enumeration	One of a set of items (e.g. “apple”, “orange”, “banana”).
Integer	A whole number (e.g. 5)
String	Text (e.g. “The sixth sick sheik”).

In our example, the data type would be *Double*. If we choose “enumeration”, a box will appear below the *Data type* selection. In this box, enter the items to be enumerated, one per line.

The *Visualization* button is used to define how the field will be displayed in the map. This is discussed in greater detail in the section “Displaying properties” below.

Setting properties

Now that we have set up a field, we probably want to set some properties using the field. There are two basic ways of doing this: using the Properties dialog box, or as a side-effect of the project editing tools.

Using the Properties dialog box

The Properties dialog box for many layers contains a page also called *Properties*. This is used to edit flexible properties manually. At the bottom of the page is an *Add Field* button. This can be used to add fields without having to exit the Properties dialog box. It is a short-cut to the *Field Properties* dialog box described above. The remainder of the page is composed of a series of property definitions, one per line. Initially, there is just one blank line, as shown in Figure 26.

Figure 26 - A blank property line

If we click on the drop-down at the left, we will see a list of fields we have defined. When we choose a field from this list, the blank space to the right becomes a set of controls for defining the property. For example, if we select the “hole depth” field defined above, we get the property line shown in Figure 27.

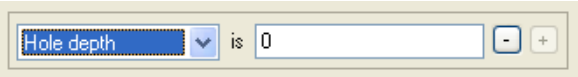


Figure 27 - A typical property line

We can enter a value for the property in the box. The contents of this area will depend on the field. For an enumeration field, for example, it will be a drop-down list of items. For a Boolean field, there is no data at all—the existence of the property indicates a "true" value.

To the right of the property settings are - and + buttons. These are used to delete a property or insert a new one. When you have finished setting properties, click *OK* to close the Properties dialog box.

Using project editing tools

The most common way of changing a layer's properties is as a side-effect of using one of the project editing tools. For example, suppose we have built a theoretical survey and have obtained air photos. Our next step might be to move some lines to existing trail. It would be handy if the tool could mark the rechained sections as "existing" so that we can use this information in our reports later on. Several of the tools have a *Properties* button for this purpose.

Suppose, as mentioned above, that we want to mark rechained sections of line as "existing". First, we need to set up a Boolean field called "Existing", as described in "Managing fields" above. Next, bring up the *Rechain Tool*. Click the *Properties* button in the tool. This will bring up the *Flexible Properties* dialog, shown in Figure 28.

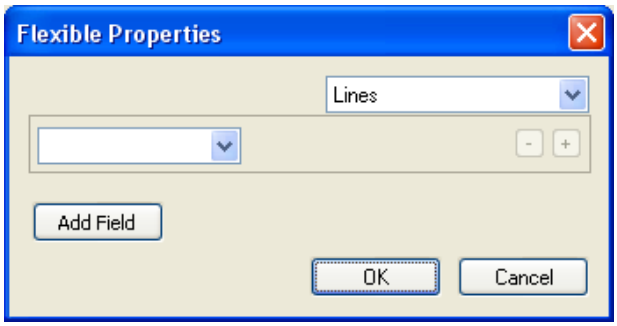


Figure 28 - The flexible properties dialog

We can set the properties for the new line created by the tool, the stations moved, or both. To view the property settings for lines or stations, choose from the drop-down list in the top-right. If a property is not listed in this dialog box, then it will remain unchanged by the tool. In this case, we would choose "Lines" from the drop-down list at the top, and choose the "Existing" property from the drop-down list on the left. This will ensure that affected line segments are given the "Existing" property.

Click *OK* to return to the Rechain Tool, and make some changes. Presumably the properties are being set behind the scenes, but we can't *see* what's been marked. This is discussed in the next section.

Displaying properties

There is little point in marking properties if we can't see what's been marked. To change the way that a field is visualized, choose *Manage Fields* from the *Tools* menu. The *Manage Fields* dialog box, shown in Figure 29 is displayed.

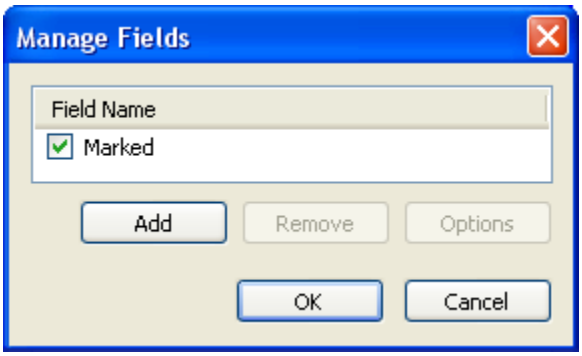


Figure 29 - The Manage Fields dialog

To turn field visualization on or off, simply check or uncheck the box to the left of the field description. To change visualization options for a particular field, choose the field from the list and click the *Options* button. The Options dialog is useful for two reasons. First, if you forget what the reference name for a field is, you can check it here. Second, you can change the visualization options for a field.

To change visualization options, select a field from the list and click the *Visualization* button in the bottom-left corner of the Options dialog. Initially it looks pretty empty, as shown in Figure 30.

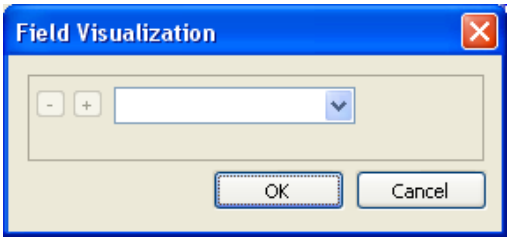


Figure 30 - An empty visualization dialog

Visualization options can be applied to points and/or lines. Continuing the example above, suppose we want to mark existing line with a green highlight. First, in the *Manage Fields* dialog, choose the "Existing" property from the list and click *Options*. Now click the *Visualization* button. Choose "Lines" from the drop-down at the top of the dialog box. The dialog box should change as shown in Figure 31.

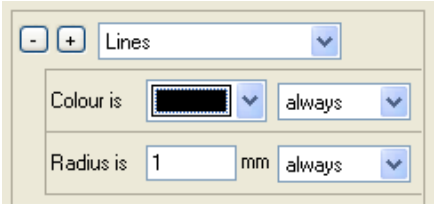


Figure 31 - Line visualization options

Choose green from the *Colour* drop-down. In this case, we actually want the line segments to be green *always* if they have the "Existing" property. Set the radius to 3 mm or so. Any section of line marked with the existing property should now look like Figure 32.

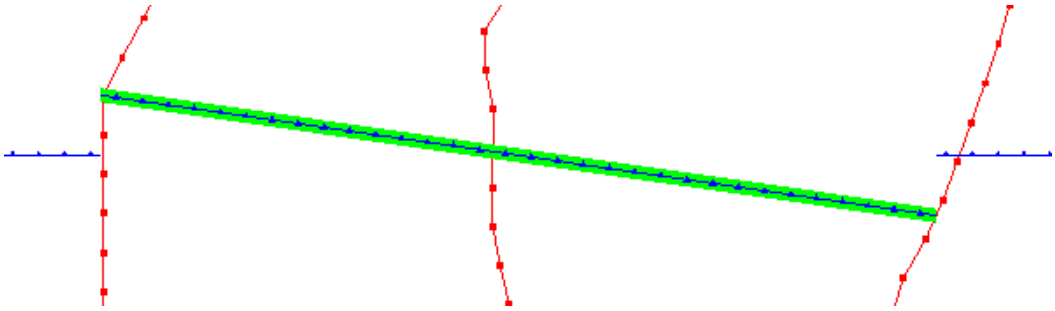


Figure 32 - A section of marked line

We have used the simplest possible set of rules for field visualization, but the rules can be much more complex. Suppose we wanted to visualize hole depth. Let's set up hole depth so it's shown as a circle with radius depending on the depth. We assume that you have already created a field called "Hole depth" with data type "double", and that you have marked some sources with their hole depth. Next, in the *Manage Fields* dialog, choose the "Hole depth" property from the list and click *Options*. Now click the *Visualization* button. Choose "Points" from the drop-down at the top of the dialog box. The dialog box should change as shown in Figure 33.

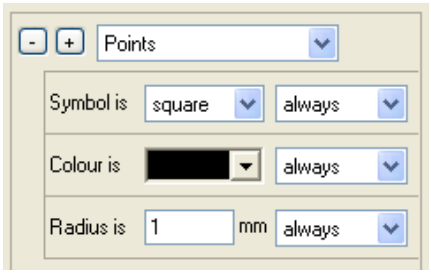


Figure 33 - Point visualization options

Set the symbol to "circle always". Set the colour to orange "always". For the radius, make the changes shown in Figure 34.

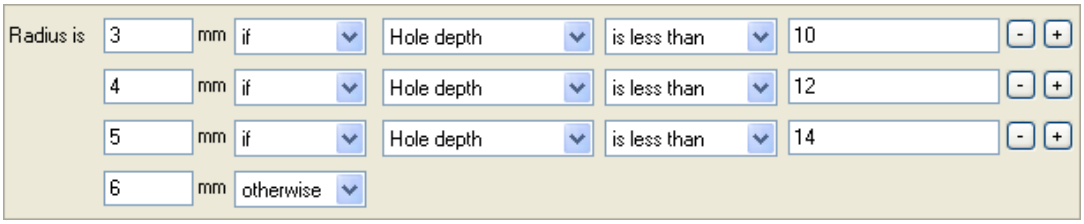


Figure 34 - A variable radius

These rules are evaluated from top-to-bottom. The result is that the radius of the circle depends on hole depth, as shown in Figure 35. Of course, the actual threshold values for hole depth will depend on the range of your values. The same trick can be used to make symbol or colour dependent on a property's value.

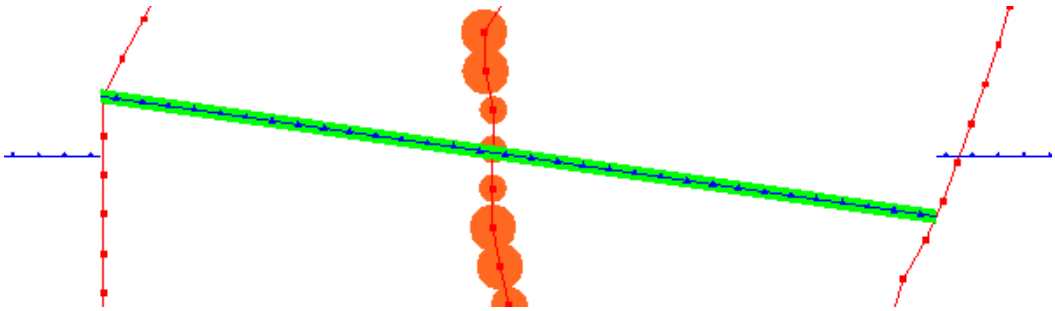


Figure 35 - Hole depth visualization

Filtering with properties

Using properties, it is possible to choose which shots are included in a generated template. This would be useful, for example, if you were using more than one shooting method in a project. In this case, you might want to shoot one set of shots with Patch "A", and another set with Patch "B". Suppose the majority of shots use Patch "A", and only a few use Patch "B". Then you might mark the latter with a Boolean property, say, "Uses Patch B". When it comes time to generate scripts, click the *Filter* button in the bottom-left corner of the Script Wizard. This will bring up the *Filter* dialog box, shown in Figure 36.

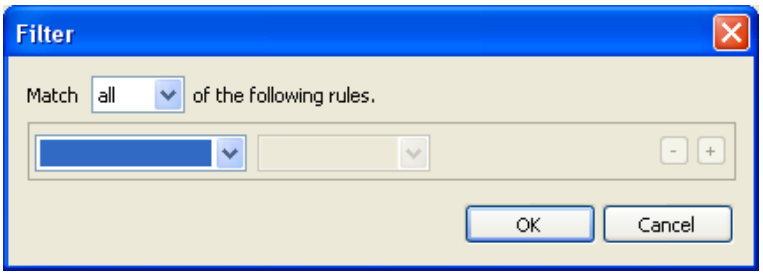


Figure 36 - The filter dialog

To modify the filter, make a selection from the highlighted drop-down list. In the example, we would select “Uses Patch B”. The result is shown in Figure 37. The default rule, “is true” is exactly what we want, so no further changes are necessary.

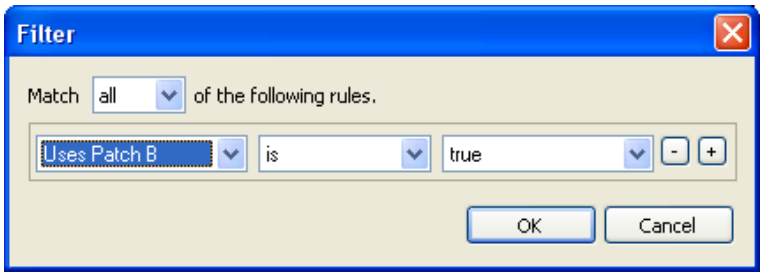


Figure 37 - A simple filter

For the second script file, we would again apply a filter, but this time we would modify the rule to read “is false” instead.

Equation constraints

If you are filtering an “integer” or “double” field, one of your filter options is “*x* satisfies *y*”. The field to the right must contain an equation in terms of *x*, where the variable *x* stands for the value of the field for each station. The following symbols can be used:

const	constant value (e.g. 1, 2, 3, 3.14)
x	field value
-	negative value
+	addition
-	subtraction
*	multiplication
/	division
%	modulus (remainder)
<	less than
>	greater than
<=	less than or equal to
>=	greater than or equal to
=	equal to
!=	not equal to

For example, the equation “x % 2 = 0” will select all the stations with an even field value. The equation “x % 2 = 1” will select all the stations with an odd field value.

Grids

Grids in DirectAid are map layers, and can also be used to guide cursor movements. They fall into three basic categories:

- *Regular grids* are a series of regularly spaced lines in some coordinate system. UTM grids and latitude/longitude grids are examples of regular grids. *Project grids* are a special case of regular grids.
- *Regional grids* are usually specific to a particular region. They generally have more structure than a regular grid, and may be based on actual surveyed points. The Dominon Land Survey (DLS) and National Topographic System (NTS) are examples of Canadian regional grids.

Except for the project grid, all grids are created by right-clicking in the blank area of the layer view and choosing *New/Grid Layer* from the context menu. The first page of the *New Grid* wizard asks what type of grid you would like to build. Depending on the type of grid you specify, the following pages will ask for more detailed information.

Project grids are created by right-clicking on a project in the layer view and choosing *New/Grid Layer* from the project's context menu. DirectAid will create the grid as a child layer of the project. By default, the grid spacing is equal to the line spacing.

Project grids

Project grids are a special case of regular grids where the grid spacing is some multiple of the line spacing. To change the multiplier, right-click on the grid layer and choose *Properties* from the context menu. The *Properties* dialog box contains two tabs: one with properties specific to the project grid, and the other with line styles for the grid. The *General* tab of the project grid properties dialog box is shown in Figure 38.

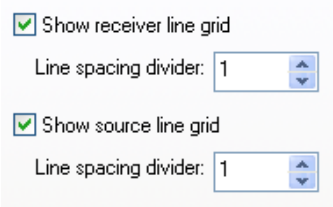


Figure 38 - Project grid properties

Using the *General* tab, you can turn on/off the receiver grid and the source grid, and also adjust the line spacing divider in each direction. By adjusting the divider, you can put additional grid lines between project lines. This is useful, for example, for accurately interpolating lines. For more information on using the project grid to guide design work, see "Project grids" on page 25.

Regular grids

To build a regular grid, right-click in the blank area of the layer view and choose *New/Grid Layer* from the context menu. On the first page of the *New Grid* wizard, choose *Basic Grid*. Click *Next*. The following three pages are the standard unit, datum, and coordinate system pages. It will be easiest to build the grid if you select a unit which is appropriate to the grid spacing. For example, if you are building a 1 km UTM Zone

10N grid using the NAD27 datum, choose “Kilometer” as your linear unit, “NAD27” as your datum, and “UTM Zone 10N” as your coordinate system.

The last page contains options specific to a regular grid. The basic version of this page is shown in Figure 39.

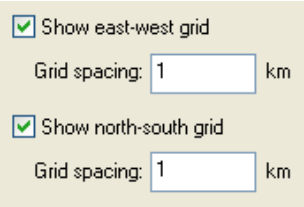


Figure 39 - Basic options for a regular grid

This makes it easy to create the most common type of grid. In some cases, you may want to create a more complex grid. This can be done by checking *Advanced options* in the lower-left corner of the page, which yields the page shown in Figure 40.

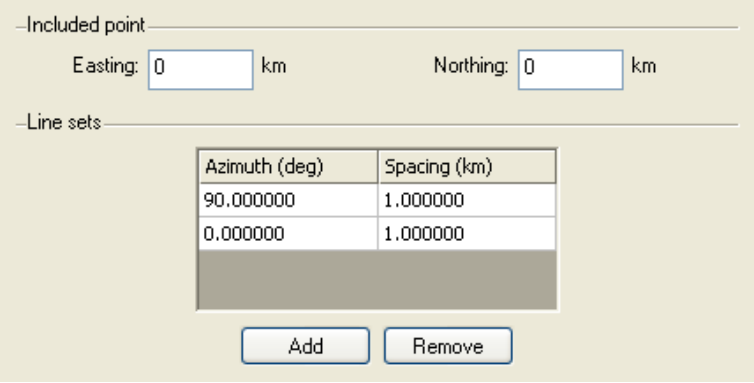


Figure 40 - Advanced options for a regular grid

On this page, you are asked to specify an included point and one or more sets of line parameters. Each set of line parameters defines a set of regularly spaced parallel lines. The included point is the point through which one line from each set passes.

When you have set parameters for the grid, click *Finish* to continue to the final step in creating a grid. In this step, the *New Grid* wizard disappears and the *Move Grid* tool (discussed in greater detail on page 29) is activated. Use the *Move Grid* tool to define an area over which the grid is valid. When you have defined the grid area, close the *Move Grid* tool.

DLS grids

The Dominion Land Survey is the method used to divide western Canada into one square mile sections. It is the dominant survey method for the prairie provinces, but is also used

in parts of British Columbia. Each section (Figure 41) is further divided into 16 legal subdivisions (Figure 42).

Figure 41 – The DLS grid

Figure 42 – Legal subdivisions

To use a DLS grid in DirectAid, you will require additional third-party "grid" files which define the northeast corner of each section. Currently, the only files which are supported by DirectAid come from Terra Management. Several versions of the grid are available, so if you are interested in purchasing a copy, please contact us first. If you use another set of files and would like to see these supported by DirectAid, please let us know.

Terra Management offers several versions of the DLS grid, including ATS 2,2, ATS 2.6, ATS 3.1, ATS 3.2, MTS, and STS 1.0. These are usually stored in separate folders. To use these files in DirectAid, you will need to copy these folders into a folder called "dls" in your DirectAid program folder. Your directory structure should look something like Figure 43.

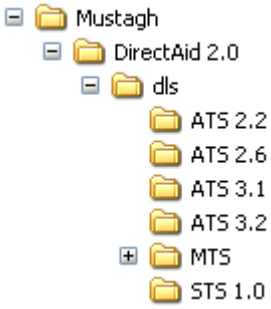


Figure 43 – DLS folders

The names of the individual DLS folders will be used by DirectAid to identify each grid, so it is worth spending a moment to make them look nice. Once the data is in the right place, you are ready to use DLS grids in DirectAid.

To build a DLS grid, right-click in the blank area of the layer view and choose *New/Grid Layer* from the context menu. On the first page of the *New Grid* wizard, choose *DLS Grid*. Click *Next*. The following page asks which DLS grid you would like to use. The list is populated with the names of sub-folders in the "dls" folder. Choose a grid version and click *Finish* to continue to the final step in creating the grid. In this step, the *New*

Grid wizard disappears and the *Move Grid* tool (discussed in greater detail on page 29) is activated. Use the *Move Grid* tool to define an area over which the grid is valid. When you have defined the grid area, close the *Move Grid* tool.

By default, DLS grids are drawn down to the section level. By right-clicking on the grid in the layer view and choosing *Properties*, you can change drawing and labeling options for the grid.

NTS grids

The National Topographic System of Canada provides topographic map coverage of Canada. The NTS grid describes the boundary of 1:250,000 (e.g. 82J) and 1:50,000 (e.g. 82J14) map sheets. In addition, the system has been extended by dividing each 1:50,000 map sheet (Figure 44) into 12 lettered “blocks” (Figure 45) and 100 numbered “units” (Figure 46). The boundaries of older NTS map sheets were defined using the NAD27 datum; newer map sheets are defined using the NAD83 datum.

Figure 44 - The NTS grid

Figure 45 - The lettered sub-grid

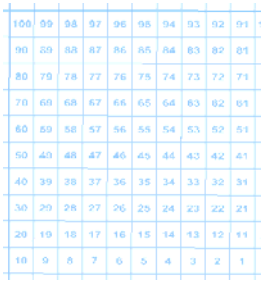


Figure 46 - The numbered sub-grid

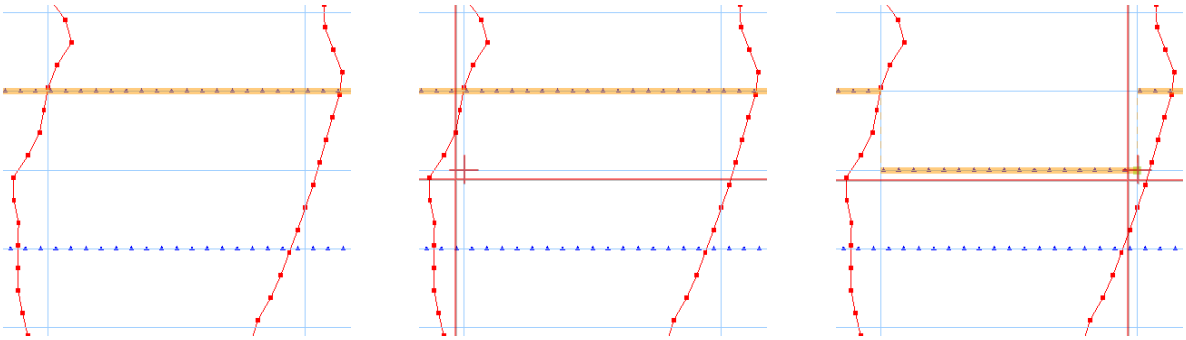
To build an NTS grid, right-click in the blank area of the layer view and choose *New/Grid Layer* from the context menu. On the first page of the *New Grid* wizard, choose *NTS Grid*. Click *Next*. The following page is the standard datum page, except that it has been restricted to NAD27 and NAD83. Choose a datum and click *Finish* to continue to the final step in creating the grid. In this step, the *New Grid* wizard disappears and the *Move Grid* tool (discussed in greater detail on page 29) is activated. Use the *Move Grid* tool to define an area over which the grid is valid. When you have defined the grid area, close the *Move Grid* tool.

By default, NTS grids are drawn down to the 1:50,000 map sheet level. By right-clicking on the grid in the layer view and choosing *Properties*, you can change drawing and labeling options for the grid.

Snap to grid

In many tools, it is possible to constrain cursor motion to points on a grid by holding the *Ctrl* key in the middle of an editing operation. The following series of pictures shows snap-to-grid being used in combination with the *Rechain Line* tool to move a portion of

the line exactly halfway between two theoretical line locations. In this case, we are using a project grid (see “Project grids” on page 25) with the spacing divider for receiver lines set to 2.



Note that a small “snap-to-grid” cursor appears at the grid point when the *Ctrl* key is held. This is where the vertex will actually be created.

Using vector layers as grids

In addition to “native” grid layers, it is also possible to use a vector layer as a grid. To do this, right-click on the vector layer in the layer view and choose *Use As Grid* from the context menu. The cursor should now snap to vector layer objects as well as grids when you hold the *Ctrl* key. To stop using the vector layer as a grid, right-click on the vector layer in the layer view and choose *Use As Grid* again from the context menu.

By using a vector layer as a grid, the snap-to-grid feature can be used to rechain the lines of a project to follow lines in a vector layer. For example, you may have received a DXF file detailing the location of existing trail. In this case, you could load the DXF into DirectAid, choose *Use As Grid* from the new layer’s context menu, and then switch to the *Rechain Lines* tool. By holding the *Ctrl* key while choosing points for the rechained line, you can constrain these points to lie along a grid—in particular, the DXF file you just read in.

Moving grids

When you initially create a grid layer, you will need to define an area over which the grid is drawn. This rectangular area is defined using the *Move Grid* tool. Grid boundaries can also be modified after the grid has been created by right-clicking on the grid in the layer view and choosing *Move Grid* from the context menu. This will activate the *Move Grid* tool.

If a bounding box has not already been defined for the grid (for example if you have just created the grid), then the grid will not be visible in the map view. You can make the grid visible by clicking and dragging with the left mouse button in the map view. This will define a box with one corner at the point where you press the mouse button, and the opposite corner where you release the mouse button. If you do not see the grid inside this

box, the most likely cause is that your grid spacing is too large—there are no grid lines within the box. If this is not the case, double-check the units, datum, coordinate system, and grid spacing (paying particular attention to the units used to define the spacing), and try the *Move Grid* tool again.

If a bounding box has been defined for the grid, then there are two ways to use the tool. The first is to click and drag in the map and redefine the bounding box. The second is to click and drag on the vertices of the existing box to adjust it. When you have finished setting the grid's bounding box, close the *Move Grid* tool.

Reports

In addition to the ability to print maps directly to a printer (see "Printing and Print Preview" on page 86), DirectAid features flexible reporting based on the popular Python programming language (www.python.org). We have chosen to use Python because it is easy to learn and has tremendous community support.

Our goal for reporting in DirectAid has been to give the user complete control over how the information stored in the workspace is manipulated and presented to build the report. To achieve this, we have given Python complete access to workspace data. Reports are generated entirely using Python scripts, which in turn make use of open source third-party libraries to incorporate elements such as images and PDF documents in the report.

This may sound very complex to users who are not themselves programmers. However, we have included a few basic reports which will suffice for most users' needs. As you will see in the following section, applying these reports is very easy. If you find that these reports do not meet your requirements, please let us know and we will help you create what you are looking for.

Additional information for users who are interested in writing their own reports can be found in "DirectAid Extensions to Python" on page A-1.

Applying reports

Active layers

Because a single workspace may contain multiple projects, before you can generate a report you will need to tell DirectAid which project to report on. This is done by setting an "active layer". Active layers are marked as bold in the layer view, as shown in Figure 47.

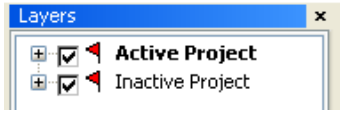


Figure 47 - An active project

To set an inactive project as the active layer, right-click on the project in the layer view and choose *Active Layer* from the context menu. If the menu item is checked, then the layer is already active.

Each project has one "active" midpoint layer. The active midpoint layer is the focus of attention in reports which include subsurface plots such as fold. If a report includes a colour scale, this scale refers to the active midpoint layer. If a report manipulates the current midpoint plot (for example, changing it to a detailed offset distribution plot), then it is the active midpoint layer which will be affected.

The active midpoint layer is shown in bold in the layer view. To set a midpoint layer active, right-click on the midpoint layer in the layer view and choose *Active Midpoint* from the context menu. If the menu item is checked, then the midpoint layer is already active. If a midpoint layer is active and visible, you should see a midpoint layer legend in lower-right corner of the map view.

Generating the report

Once you have selected an active project and an active midpoint layer, you can generate any report with those two layers as the "target" of the report. First, switch to the report view by clicking on the "Reports" tab at the bottom of the layer view. This will display a list of available reports, as shown in Figure 48.

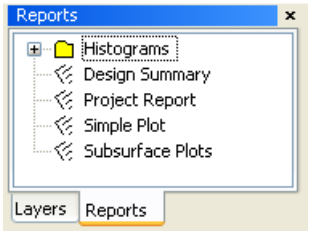


Figure 48 - The report view

To get a description of the report, hold the mouse cursor over the report for a moment. This will display a balloon including the name of the report, a description, and an image showing a typical report, as shown in Figure 49.

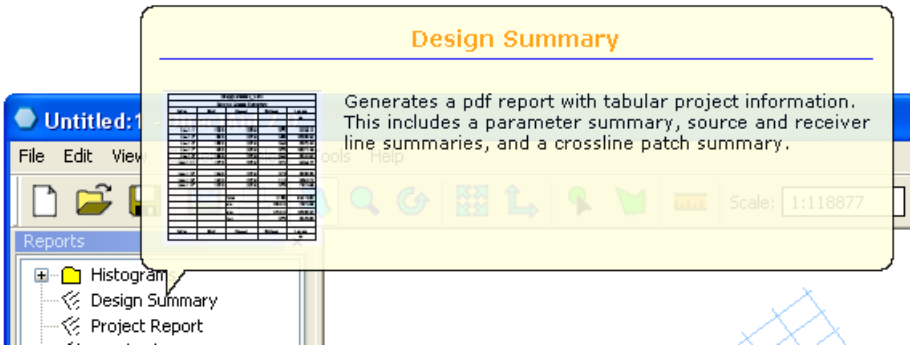


Figure 49 - A report description

To generate a report using the current active layers, right-click on the report and choose *Run* from the context menu. The first time you generate a report, it may take a few moments to complete. When the report is complete, it will open a window showing you the generated report. If the report is a PDF, DirectAid will open Acrobat Reader with the completed report.

If the report generates errors, DirectAid will bring up an error dialog box like the one shown in Figure 50 below.

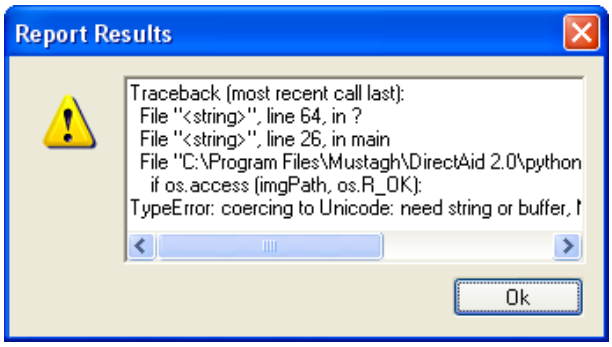


Figure 50 - The report error dialog box

Although the diagnostic information displayed is really only useful to those who are familiar with the Python programming language, the two most common causes of these errors are:

- No active project has been defined. Right-click on the project and check that *Active Layer* is checked in the menu. If it is not checked, click on *Active Layer*.
- The file being generated by the report is already open. This can happen if you've generated the report once before and have not closed that window.

Reports included with DirectAid

As mentioned above, several reports are included with DirectAid. These reports are intended to take care of basic plotting and reporting needs, and as samples for more advanced users who wish to build their own reports. The following table lists the reports included with DirectAid:

Report Name	Description
Design Summary	A tabular PDF report giving: • Parameter summary • Source and receiver summaries • Crossline patch summary
Project Report	A package including the design summary, the fold histogram, and subsurface plots.
Simple Plot	Generates a PDF with a map of the current view and labels.
Subsurface Plots	A package of plots in a single PDF on letter size paper. Camera and midpoint layer settings are modified to yield the following output: • Fold plot • Offset-squared gap deviation • Offset-squared distribution (detail) • Azimuth gap deviation • Azimuth distribution (detail) • Midpoint distribution (detail) • Offset-azimuth distribution (detail)

In addition to these reports, several histograms are included:

Histogram Name	Description
Fold Histogram	A histogram showing the distribution of fold in subsurface bins.
Offset-Squared Gap Deviation	A histogram showing the distribution of offset-squared gap deviation in subsurface bins.
Azimuth Gap Deviation	A histogram showing the distribution of azimuth gap deviation in subsurface bins.
Offset-Azimuth Distribution	A two-dimensional histogram showing the distribution of offset and azimuth in subsurface midpoints.

Scripts

Building scripts

To build scripts for a project, right-click on the project in the layer view. From the context menu, choose *New* and then *Script layer*. This will bring up a sheet asking for details on the shooting method. Five shooting methods are currently available. To choose between these methods, select from the drop-down list in the bottom-right corner of the sheet, shown in Figure 51.

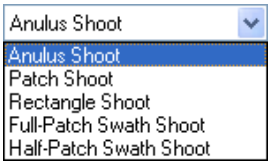


Figure 51 - Shooting methods

Annulus shooting

An annulus shoot is defined by a minimum and maximum radius from the source. If the minimum radius is set to “0.0”, then this is a circle shoot, including every receiver out to the maximum radius.

By clicking the *Advanced parameters* checkbox at the top-left of the page, you can change two additional parameters. The annulus can be offset from the source by a specified distance in the inline and crossline directions. This is not generally recommended, but is included for advanced shooting methods.

Patch shooting

A patch shoot is defined by a certain number of lines and a certain number of stations on each of those lines, usually centered on the source point. Options for the patch shoot are shown in Figure 52.

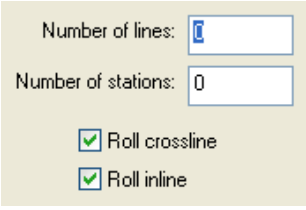


Figure 52 - Patch shooting options

In DirectAid, the patch shoot is actually defined by a certain number of lines and a range of inline distances, centered on the source. Usually this is similar to specifying a number of stations on each line (simply multiply the number of stations by the interval to get the

inline patch size), but the new method deals very well with gaps in lines and extra stations.

You also have the option to *roll crossline* or *roll inline*. Roll simply means that when the patch hits the edge of the survey, it drops lines or stations to remain centered on the source. You will usually create patches with both of these options enabled. The most common case where you would not use inline roll is if you are shooting with every station on a line live. Similarly, you will likely disable crossline roll only if you are shooting with every line live.

By clicking the *Advanced parameters* checkbox at the top-left of the page, you can change several additional parameters, as shown in Figure 53.

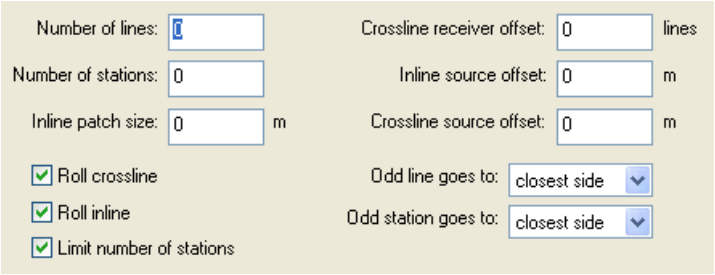


Figure 53 - Advanced patch shooting options

If there are gaps in the line, the patch will probably include fewer stations than specified. If there are extra stations in the line, this will actually result in more stations in the patch than you might have expected. This is done to maintain a consistent offset range in the patch, since the patch would not extend to long offsets without additional stations. If you would like to restrict the patch so it will never use more than the specified number of stations, check the box labeled “Limit number of stations”.

There are two methods of offsetting a patch. First, you might pretend that the source was actually in a different place, and generate patches as normal from there. This is known as *source offset*, and can include inline and crossline components, measured in metres. Alternatively, you might shoot the patch with the source in its true position, but rather than selecting the same number of receiver lines on either side of the source, you might select more on one side. This is known as a crossline receiver offset, and is given as a number of receiver lines.

Finally, in general, it is recommended that you shoot an orthogonal survey with an even number of lines in the patch. Assuming your source lines are not coincident with receivers, you should also be shooting with an even number of stations in the patch. However, if you choose to shoot with odd numbers in these circumstances, there is some uncertainty as to which side of the source the “extra” receiver line or station should go to. This is resolved by the options labeled “Odd line/station goes to”.

When you have entered all the parameters for the script, click *Finish* to complete the operation. DirectAid will generate the scripts, add them to the project in the layer view, and the *View Templates* tool will be activated.

Rectangle shooting

A rectangle shoot is defined by an inline (parallel to the receiver lines) and crossline (perpendicular to the receiver lines) patch size. By default, the patch is centered on the source. Receivers within the box are included in the patch, and those outside the box are not.

If you choose to use a rectangular patch, there are a couple of related issues you should be aware of. First, the receivers included from a single line may not form a single contiguous range. That is, there may be a "gap" in the patch on a given line, if the line passes out of the patch and back into it. Second, the patch may include only a small number of stations from a line near the boundary of the patch, if the line lies mostly outside the patch, but passes into it briefly. These two situations are pictured in Figure 54 and Figure 55.

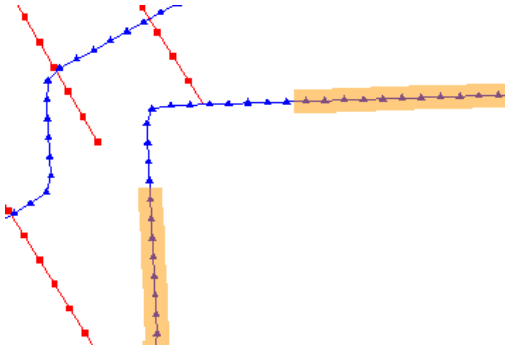


Figure 54 - A gap in a rectangular patch

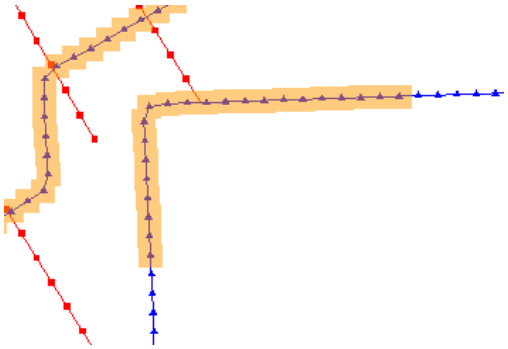


Figure 55 - An accidental inclusion

By clicking the *Advanced parameters* checkbox at the top-left of the page, you can change two additional parameters. The rectangle can be offset from the source by a specified distance in the inline and crossline directions. This is not generally recommended, but is included for advanced shooting methods.

Swath shooting

A swath shoot is similar to a patch shoot, except that more than one source swath can be associated with a single patch. A swath shoot may be either full-patch—in which case the entire patch is included in each template—or half-patch—in which case only half the patch is included in each template. The two types of swath shoot are pictured in Figure 56 and Figure 57.

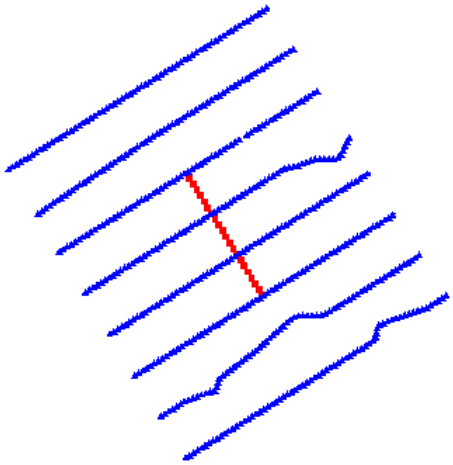


Figure 56 - Full-patch swath shoot

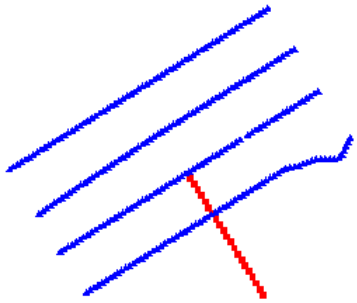


Figure 57 - Half-patch swath shoot

The half-patch swath shoot can be used to generate scripts for the "ping-pong" shooting method, in which two sets of vibrators are used—one shooting from one side of the patch, and the other from the opposite side.

Numbering the swaths

Swath shooting in DirectAid is a two-step process. First, you will need to number the source swaths. Second, you will create a script using these swaths. This has been done to give users extra flexibility in cases where the definition of the source swaths is not completely obvious—for example, when stub receiver lines are involved.

To number the source swaths, right-click on the "Sources" layer in the layer view. From the context menu, choose either *Number Full Swaths* or *Number Half Swaths*. A full swath is defined as all the stations between two adjacent receiver lines; a half swath is all the stations which are closest to a single receiver line. These are pictured in Figure 58 and Figure 59.

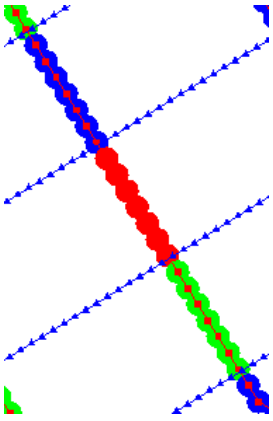


Figure 58 - Full swaths

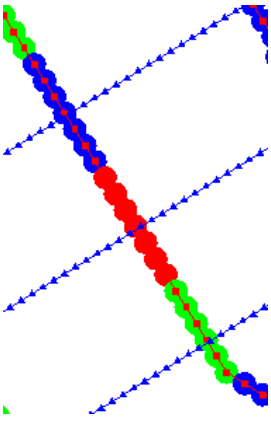


Figure 59 - Half swaths

The decision to use full swaths or half swaths depends on how many receiver lines and source swaths are in your patch:

	Odd number of lines	**Even number of lines**
Odd number of swaths	Use half swaths	Use full swaths
Even number of swaths	Use full swaths	Use half swaths

Note that if you are using a half-patch swath shoot, you'll need to use the number of lines in the complete patch, that is, twice the actual number of lines in the patch, when doing this calculation.

Automatic swath numbering sets the "_Swath_Number" property for each source, based on survey geometry. In some cases, this may be a little different from what you expect. Take, for example, the region shown in Figure 60.

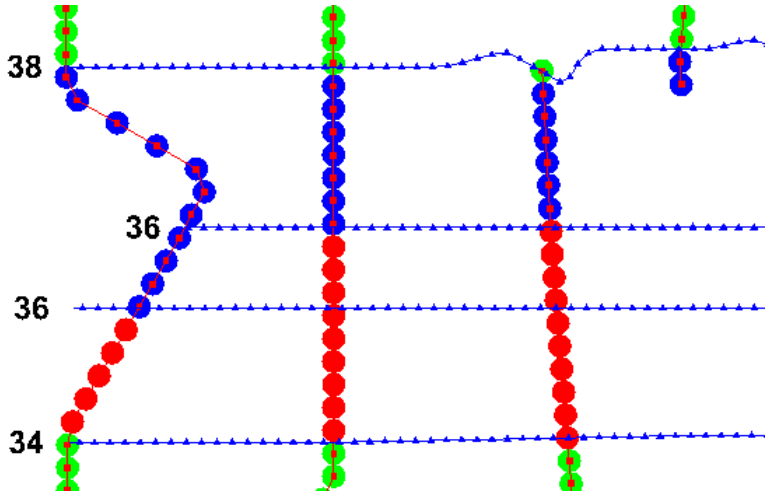


Figure 60 - An incorrectly numbered swath

Here, we are looking at a stubbed receiver line. Line 36 runs from east to west. The main line is the north one, but at the west end of that line, a stub has been added to the south. When calculating swath numbers, DirectAid ignores the smaller stub line, except where it finds no intersection with the main line. In this case, you can see that it has arguably mis-numbered a few shots on the west-most source line. We can fix these manually by changing the "_Swath_Number" property for those stations.

Creating the script

Once you're satisfied with the swath numbering, you're ready to create scripts. Right-click on the project in the layer view. From the context menu, choose *New* and then *Script layer*. From the list of shooting methods, choose either "Full-Patch Swath Shoot" or "Half-Patch Swath Shoot".

The parameters on the left side of the window define the size of the patch, as well as roll. If you are using a half-patch swath shoot, the number of lines is the actual number of lines which will be live, and not the total number of lines in the "full" patch (which is twice that number).

On the right side of the window, both swath shooting methods ask for "Number of swaths" and "Swath adjustment". The first parameter is the number of adjacent swaths which will use the same receiver patch. For example, Figure 56 shows a patch with 3 swaths. By default, swaths 0 through $N-1$ will share the same receiver patch, as will swaths N through $2N-1$, and so on. You can use the swath adjustment to bump these ranges forward or backward.

In addition, the half-patch swath shoot needs to know which half of the patch you want to use for this script. If you're writing scripts for ping-pong shooting, each source will be shot twice—once by each set of vibrators. One set of vibrators will shoot with the swath at the high end of the patch, and one with the swath at the low end.

Viewing scripts

It is highly recommended, after creating or importing scripts, that you take a moment to check them using the *View Templates* tool. This tool is activated automatically after creating scripts, but can also be activated by choosing right-clicking on a script layer and choosing *View Templates* from the context menu. The tool is shown in Figure 61.

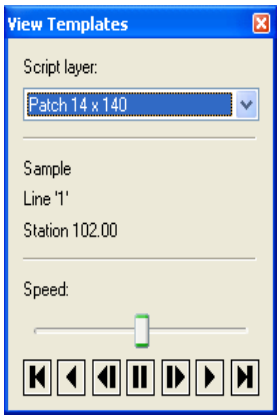


Figure 61 - Viewing templates

To use the *View Templates* tool, first select a script layer from the drop-down list at the top. The buttons at the bottom of the tool are used to step through sources in the parent project:

- Go to first template
- Play backward
- Step backward
- Pause

- Step forward
- Play forward
- Go to last template

After selecting a template, usually the first thing you will do is to go to the first template. Now, press the “play forward” button. DirectAid will automatically step through templates. To control the speed of the animation, use the slider in the tool. With the slider centered, the animation is paused. To the left, the animation moves backward; to the right, it moves forward. If you have a mouse wheel, you can use it to zoom into the map and pan around while the animation proceeds. This can be used to check possible problem areas in greater detail.

Finally, you can check the template for a particular source point by clicking on that source in the map view. The animation, if running, will be paused.

Reading/writing script files

Reading

To read script files in SPS, ARAM project, or I/O binary format, right-click on the project in the layer view. From the context menu, choose *Import* and then *Script layer*. Using the file dialog, select the script files to be imported, and click *Open*. The scripts will be added to the project in the layer view. As always, it is a good idea to view templates after importing a script file.

Writing

To write script files in SPS, ARAM project, or I/O binary format, right-click on the script in the layer view. From the context menu, choose *Export*. In the *Save As* dialog, choose the format of the script at the bottom, enter a name, and then click *Save* to complete the operation. Depending on the format of the script file you are writing, DirectAid will ask for additional information needed for export.

For SPS files, DirectAid will need to know whether you would like to include the line label as a prefix to the station number. By default, this is not selected. You will also be asked to specify the maximum number of templates per file and overlap between files. The default value of 3600 templates per file should be sufficient for most systems. Following this page, you will find the three coordinate system pages. Although the SPS script file itself doesn’t require coordinate system information, the header of the file does. You should use the same selections here that you use when writing out the SPS point files.

For ARAM project files, you will be asked what software version will be used to read the file. This refers to the ARAM software version. If you are using integer line labels and station numbers, as is usually the case, then all three options are equivalent. Otherwise, you will need to find out what version of software is being used by the crew. Following

this page, you will find the three coordinate system pages. This is the coordinate system that will be used when writing source and receiver stations to the ARAM project file.

I/O binary scripts do not require any additional parameters.

Merging scripts

In most cases, you will use a simple script to shoot a project. In some cases, however, you may want to combine several simple scripts to build a more complex script. For example:

- To make efficient use of available equipment, you might use different patches in different parts of a survey.

- You may want to estimate the effect of using smaller charges in part of the survey. One way of doing this is to shoot the small-charge shots with a small radius patch, and the remainder of the shots with a larger radius.

In both these scenarios, you will first need to generate one “basic” script which includes every shot in the project, and a second “modified” script which includes only those shots you want to replace with a modified patch. To shoot only part of the survey, you will need to mark some of the shots using flexible properties (see “Setting properties” on page 19), and then create a script only for the marked shots (see “Filtering with properties” on page 23).

Check the scripts using the *View Templates* tool to ensure that the “modified” script is defined only for the marked shots (when you click on unmarked shots, the patch should not appear). Then drag the “modified” script onto the “basic” script in the tree view. The “basic” script should be highlighted just before you release the left mouse button.

When you release the button, you will be asked to confirm the merge operation. Click *OK* to continue. The merge process will replace individual shot templates in the target script if they exist in the source script.

Sorting scripts

When scripts for a large project are split across multiple files, it can be helpful for the recording crew if the scripts are sorted in the order they will be shot. Although DirectAid has no way of knowing what the actual shooting order is, you can use the simple script sorting options to approximate it. To sort scripts, right-click on the script in the layer view and choose *Sort Script* from the menu. This will open the dialog shown in Figure 62.

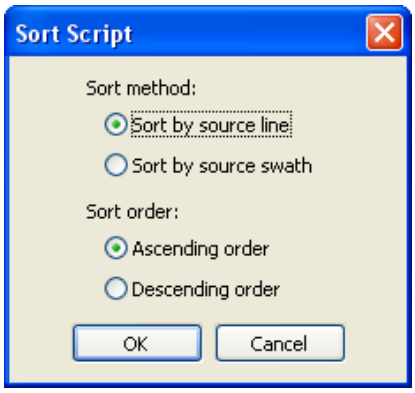


Figure 62 - Sorting scripts

Using the sort tool, you can sort either by source line or by source swath. When sorting by source line, templates on the same source line will be arranged in order of shot number.

Before sorting by swath, you must number source swaths, as described in “Numbering the swaths” on page 37.

When sorting by swath, templates in the same swath will stay in their original order. This means you can arrange templates within a swath by sorting first by source line, and then by swath.

Duplicating scripts

It is sometimes handy to have an extra copy of a script, but with a few shot templates changed. For example, you may want to merge a smaller script into a larger one, but retain a copy of the originals. To duplicate a script, right-click on the script in the layer view and choose *Duplicate* from the menu. This will create a copy of the script layer that you can change without affecting the original.

Copying scripts between projects

It is possible to copy a script from one project to another. To copy a script, drag it from the first project, and drop it on the second project in the layer view. This will create a lightweight duplicate of the script layer in the second project.

Shooting a Script

To shoot a script, right-click on the script and choose *Shoot* from the context menu. This will show the *Midpoint Wizard*. The first page of the wizard is used to specify options for generating a midpoint given a source and receiver. At this time, you may choose to use elevation data from the stations, or not. For more information on this option, see “Elevations in shooting” below.

In most cases, you will not use station elevation information. This will result in midpoints which are located halfway between the source and the receiver. To continue to the second page of the wizard, click *Next*. This will display the *Bin Options* page, shown in Figure 63.

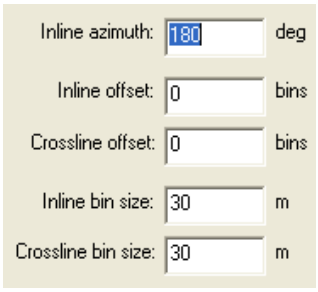


Figure 63 - Bin options

The information in this dialog box is determined automatically from the project geometry. In some cases, you may need to "tweak" the bin azimuth or position in order to keep midpoints better aligned with the bins. In general, your bin size will be determined by design parameters. When you have verified the values and made any corrections, click *Finish* to shoot the script using the parent project.

When midpoint calculations are complete, a new "midpoint" layer will appear at the top of your project sub-tree. You will probably want to drag this layer to a position below the stations so you can view stations on top of the subsurface plots. To show or hide the new plot, click on the check box next to the layer in the layer view.

Each project has one "active" midpoint layer. This midpoint layer is shown in bold in the layer view. To set a midpoint layer active, right-click on the midpoint layer in the layer view and choose *Active Midpoint* from the context menu. If the menu item is checked, then the midpoint layer is already active. If a midpoint layer is active and visible, you should see a midpoint layer legend in lower-right corner of the map view.

Elevations in shooting

On the first page in the *Midpoint Wizard*, you can choose to include station elevation information in the calculation of midpoints, as shown in Figure 64 below.

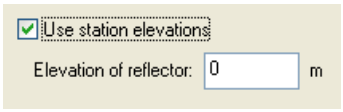


Figure 64 - Midpoint options

Usually, the reflector is assumed to sit so far below the source and receiver that the reflection midpoint is halfway between the two. However, in topographically complex

areas, and with shallow reflectors, it is sometimes useful to take into account the relative elevations of the source, receiver, and reflector.

A note of caution

Before you choose to include elevation information in the calculation of midpoints, be sure that you understand how your data will be processed. This is important because when we generate midpoint statistics such as fold, a big part of what we're trying to decide is how robust the data will be in processing. If the algorithm we use to calculate statistics is not matched with the algorithm used in processing, the results may not be meaningful.

There are two migration methods typically used in seismic data processing: pre-stack migration and post-stack migration. In pre-stack migration, the actual elevation of the stations may be taken into account. In post-stack migration, the traces are assumed to lie at the midpoint between the source and receiver. Because of this, it may be useful to take elevations into account if pre-stack migration is used, but not if post-stack migration is used.

Onward

To use this feature, you will need to have elevations assigned to your sources and receivers. At the moment, the only way to do this is if they are specified in the input file when the project is imported. In the coming year, we will be working on ways to assign station elevations from an imported digital elevation model (DEM). To make it easier to check if your elevations are assigned correctly, we have added a third field, elevation, to the position displayed when you hold the mouse over a station.

The reflector elevation uses the same coordinate system as the station elevations. If your station elevations are given as decimeters above the WGS84 ellipsoid, for example, then your reflector elevation will need to be given in the same coordinate system.

Let z_s and z_r be the height of the source and receiver above the reflector, and p_s and p_r the 2-dimensional source and receiver positions. Then the midpoint position p_m is calculated using the following pair of equations, which assume a flat reflector:

$$a = z_s/(z_r + z_s) \quad p_m = ap_r + (1-a)p_s \ .$$

Subsurface bin statistics

By default, a new midpoint layer shows a fold plot. To change the plot, right-click on the midpoint layer. The context menu displays several plot choices. Not all of these are exclusive. For example, it is possible to view the fold plot with midpoint distribution on top.

Single-valued plots

The fold plot is an example of a single-valued plot. You can have only one single-valued plot active in a midpoint layer at one time, since this is used to generate the colour background for the plot. To turn off the plot, select it a second time from the context menu.

To show the value of each bin in the map view, select *Bin values* from the midpoint layer context menu. The value will only show up at small map scales, as with detail plots (described below), and will automatically disappear when you zoom out.

Detail plots

The midpoint distribution is an example of a detail plot. Detail plots can be shown on top of single-valued plots. Detail plots do not show up at large map scales. To change the threshold scale at which detail plots appear, go to *Preferences* in the *View* menu, then select the *Project Layers* tab. Like single-valued plots, they can be turned on and off by selecting them in the context menu.

Fold

The *Fold* plot shows the total number of midpoints in each bin.

Offset and azimuth distribution

The *Offset* and *Azimuth Distribution* plots are detail plots which shows the distribution of source-receiver offsets and azimuths for midpoints contributing to each bin. Bars in both plots have a length proportional to the offset of the midpoint. In the case of the azimuth plot, this results in a “striped” appearance to the bars, since midpoints with a given azimuth may have several different offsets. Bars on the inside of the azimuth distribution plot are drawn with the highest redundancy colour. Redundancy ranges are controlled in the detail plot options.

For 3D surveys, the *Offset-Squared Distribution* is more important than the offset distribution. This is drawn with bins equally spaced in offset-squared, but with a height proportional to offset.

Offset and azimuth gap deviation

The *Offset* and *Azimuth Gap Deviation* plots represent the “smoothness” of the offset and azimuth distributions as a single value which can be viewed as a colour plot at a large scale.

To calculate this value, offsets and azimuths are arranged in increasing order. A list is then made either of the difference in offset-squared or the difference in azimuth between adjacent values. The homogeneity is the standard deviation of the differences, normalized

by the maximum possible deviation. For a perfectly even distribution, the homogeneity would be 0%, while the most uneven possible distribution would give 100%. Typical values of offset gap deviation are 4% to 20%, and for azimuth gap deviation 0% to 16%.

For 3D surveys, the *Offset-Squared Gap Deviation* is more important than the offset gap deviation. This is calculated from a list of gaps in offset-squared, and takes into account the expected surplus of long offsets compared to short offsets.

Offset and azimuth gap maximum

The *Offset Gap Maximum* and *Azimuth Gap Maximum* are single-valued plots which show the largest offset and azimuth gap, respectively.

Midpoint distribution

The *Midpoint Distribution* is a detail plot which shows the position of each midpoint within the bin. Midpoint dot size is controlled in the detail plot options.

Subsurface plot options

Many options can be set for subsurface plots. To change plot options, right-click on the midpoint layer and choose *Properties* from the context menu.

The *Colours* page controls the colour scale used for single-valued plots. The colour scale is formed by interpolating between coloured “pegs”, shown as triangles on the right side of the colour bar. Two pegs, one at either end of the scale, are fixed in place and are shown as squares. To delete and existing peg, click on the peg and hit the *Delete* key. To insert a peg, double-click to the right of the colour bar where you would like a peg to appear. Pegs can be moved by dragging them along the colour bar. The colour of a selected peg can be changed by choosing from the colour palette to the right.

The *Line Style* page controls the line style used for the bin grid. This page is described in the section “Polylines and polygons” on page 74.

To *Plots* page contains the minimum and maximum offset, as well as value ranges for each of the single-valued plots. The value ranges are shown in a single table. Cells in the table can be modified by selecting them and entering a new value, or by double-clicking to show an edit box for the value. Note the units for each of the entries, shown in the label for each row. The ranges for offset-squared values are displayed as the square root of the actual value, so that they are more meaningful for the user.

Finally, the *Detail* page controls options for each of the detail plots. To choose a plot, select it from the drop-down list in the top-right of the page.

Duplicating a midpoint layer

In some cases, you may want to keep two copies of a midpoint layer with slightly different plot settings. For example, you may want to look at the same set of midpoint data, but with different offset limits. In such a case, it would be wasteful to duplicate all of the midpoint data, which is the same for the two layers. To create a lightweight duplicate of a midpoint layer, right-click on the layer in the layer view and choose *Duplicate* from the menu. This will create a copy of the midpoint layer that you can change without affecting the original.

Editing a Project

When a project is imported into DirectAid, the program not only reads the stations, but also automatically generates smoothed polylines which represent the survey lines themselves. Thus, we have separated the notion of the line from the station. Although changes to one will result in changes to the other, so that they remain consistent, it gives us additional possibilities for editing.

Selecting lines and stations

Map objects can either be selected individually, or all objects within a specified area can be selected.

Selecting individual objects

To select individual map objects, change to map object selection mode either by choosing *Objects* from the *Select* menu, or using the toolbar button (). The *Select Objects* dialog, shown in Figure 65, will appear.

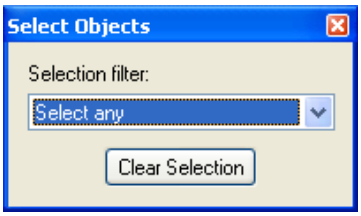


Figure 65 - Object selection tool

When your cursor is near an object which can be selected, a circle will appear around the crosshairs, as shown in Figure 66, to indicate which object will be selected. Click to select the object. To select a range of stations, click on the first station in the range and then hold the *Shift* key while clicking on the last station in the range. To add to an existing selection, hold the *Ctrl* key while selecting. The *Ctrl* key may be used in combination with the *Shift* key to add a range to an existing selection.

Figure 66 - An object marked for selection

The *Select Objects* dialog box contains a selection filter. By default, the tool will select any object. Using the filter, it is possible to restrict selection to individual stations, salvos of stations (all the stations between two line intersections), swaths (all the salvos between two line intersections), or entire lines.

Selecting objects within an area

To select objects within an area, change to area selection mode either by choosing *Area* from the *Select* menu, or using the toolbar button (). The *Select Area* dialog, shown in Figure 67, will appear.

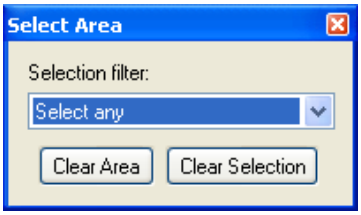


Figure 67 - Area selection tool

Click in the map view to define a selection polygon. To complete the selection, double-click on the last point of the polygon. To cancel selection, choose *Clear Area* from *Select Area* dialog box.

The *Select Area* dialog box contains a selection filter. By default, the tool will select any object. Using the filter, it is possible to restrict selection to survey lines, complete salvos of stations (all the stations between two line intersections), or individual stations.

Deleting lines and stations

Lines and stations can be deleted either in the layer view or in the map view. In the layer view, you can right-click on a single line or station, and choose *Delete* from the context menu.

To delete lines and stations in the map view, select them in the map view as described in the section “Selecting lines and stations” above. Once you have made your selection, right-click on a selected item and choose *Delete* from the context menu. If you are deleting stations, a section of line underneath the stations will be “broken”.

Moving lines and stations

The first method of editing projects is to drag lines and stations by hand. First, select the lines or stations you want to move (see “Selecting lines and stations” above).

Selection rules in the map view are similar to the rules in the layer view. To select a single object, place the cursor near the object. When the cursor is within the hit test radius (defined in the *Preferences* dialog under the *View* menu), a circle will appear around the object, as shown in Figure 68. When you place the cursor near certain objects, such as project lines or stations, you may also see a small bubble appear which gives basic information about the object.

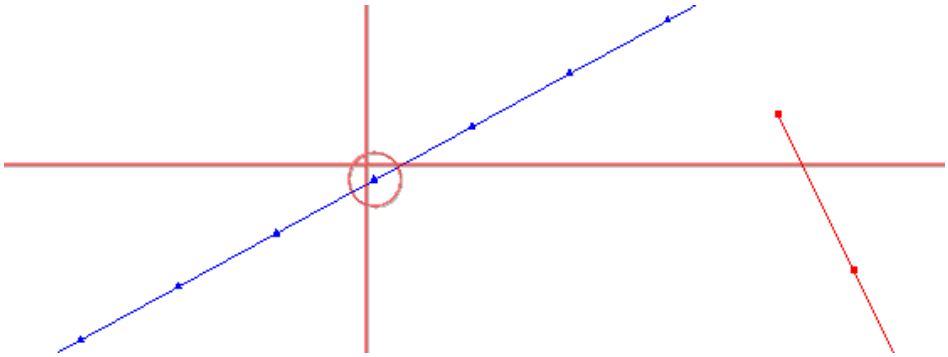


Figure 68 - The selection circle

When the selection circle is visible, clicking the left mouse button will select the circled layer. Selected layers are drawn with a transparent overlay, as shown in Figure 69.

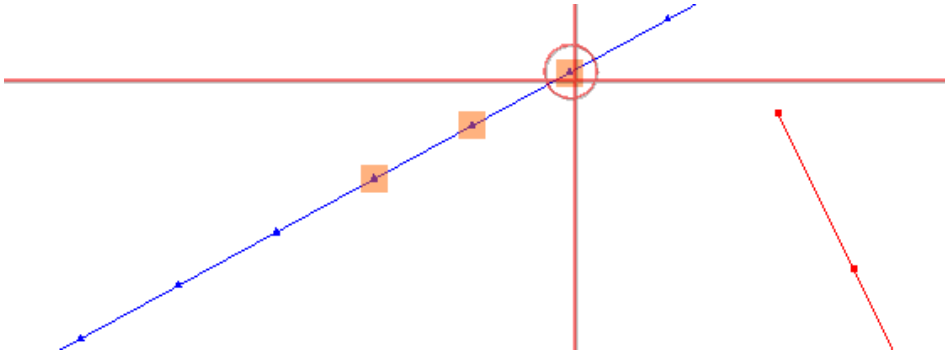


Figure 69 - Selecting a range of stations

To select a range of layers, click on the first layer, then hold the *Shift* key while clicking on last layer. To add to an existing selection, hold the *Ctrl* key while selecting. To add a range to an existing selection, hold both the *Shift* and *Ctrl* keys and select the range as usual.

To move a selected group of objects, just click and drag the selection using the left mouse button. The objects will be translated rigidly in their source coordinate system. When you

drag a group of stations, the polyline will be broken halfway between the ends of the selection and the next unselected station, as shown in Figure 70.

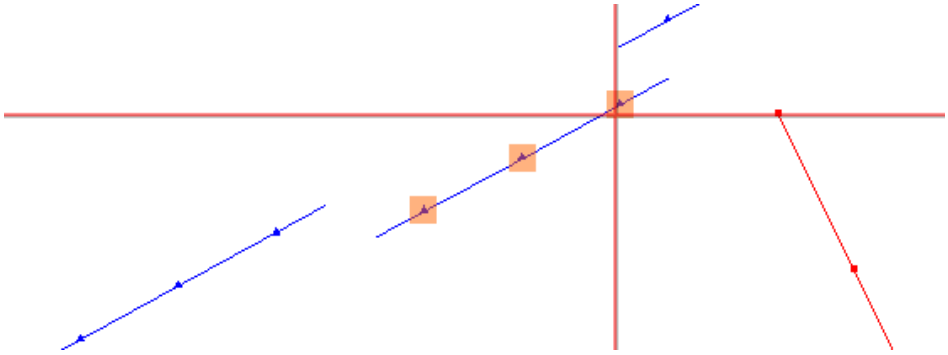


Figure 70 - Moving a range of stations

To select a project line, you can either click on the polyline itself (the selection circle will snap to the polyline if you are far enough away from the nearest station), or you can right-click on a station in the map view to access its context menu. Choose *Select line* from the context menu to select the parent line. A selected line, like any selected layer, is drawn with a transparent overlay as shown in Figure 71. To move the line, just click and drag it.

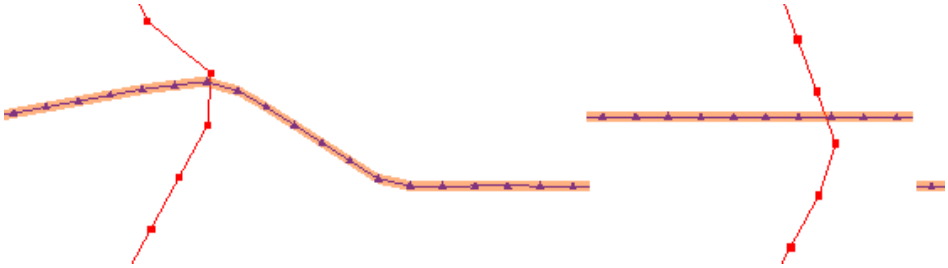


Figure 71 - Selecting a project line

The bingo card

When dragging a group of stations, it is sometimes useful to constrain movements to entire station intervals inline and crossline. As a guideline for station movement, Mustagh has developed what we call the “bingo card”, shown in Figure 72. Possible station moves are given ratings starting at “1” (not shown in the bingo card). The best position for the station is understood to be where it started. From there, crossline moves up to half-way to the next line are preferred. A crossline move one box past the half-way point is equivalent to a move one box inline and one box crossline.

To access the bingo card, hold the *Ctrl* key while dragging a group of stations. This will also constrain station movement to the center of bingo card boxes.

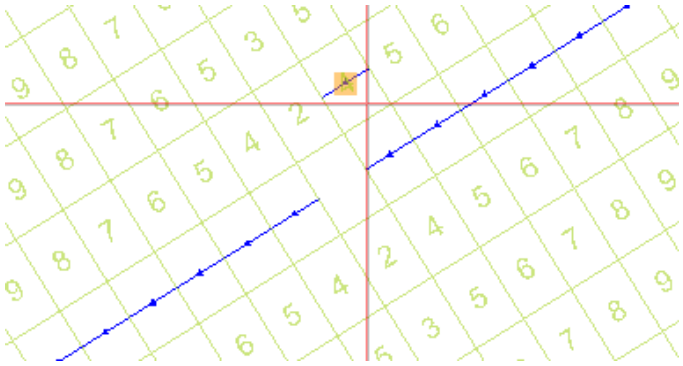


Figure 72 - The bingo card

Editing lines

The second method of editing projects is to edit the polylines themselves. To edit a polyline, select the project line using one of the methods mentioned above, then double-click on the selection. Alternatively, you can right-click on a station in the map view to access its context menu, then choose *Edit line* from the context menu to edit the parent line. When you are in editing mode, the polyline will appear with the usual transparent overlay, but will also have a thick box at each *node* of the polyline, and a thin line spanning each broken section of the polyline, as shown in Figure 73.

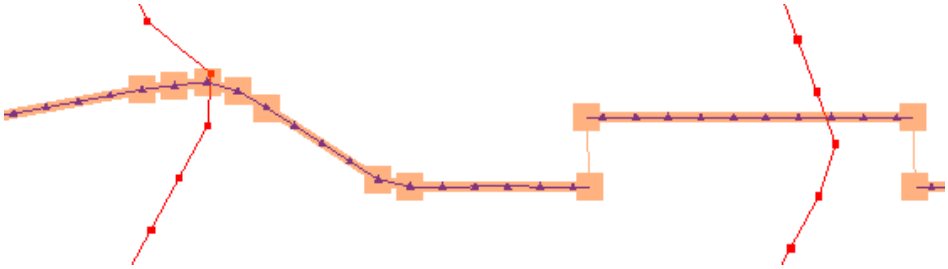


Figure 73 - Editing a project line

To edit the polyline, you can either click and drag the nodes, or create a new node by clicking on an *edge* of the polyline. As you make changes to the polyline, the project stations will follow by projecting themselves perpendicular to the survey azimuth. For example, if you are editing a receiver line, each station will be projected perpendicular to the receiver azimuth. This ensures that midpoints generated later on will remain in the same row or column of the subsurface bins.

To delete a node, right-click on the node and choose *Delete* from the context menu. To break a line segment, right-click on the segment and choose *Break* from the context menu. All stations on that segment will be deleted. To reconnect the segment, right-click and choose *Connect* from the context menu. Note that stations deleted as a result of breaking the line will not be recovered when it is re-connected.

If you are dragging a node at the end of a project line, it is possible to change the line so that one or more stations cannot be projected onto the new line. In this case, the stations will become *ghosted* as long as the mouse button is held down. If you happen to have the stations visible in the layer view, you will see their label drawn in gray rather than black. As long as the mouse button is held down, these stations remain active, although they are not visible. This means that if you drag the endpoint in so that a few stations are ghosted, and then drag it back out again, the stations will reappear. When the mouse button is released, any ghosted stations are deleted.

To exit node editing mode, simply select another layer or click on the white background of the map to deselect everything.

Snap to grid

While dragging a node, it is possible to constrain movement to the inline and crossline azimuths for the project. This can be useful for extending the end of a line, for example. To turn on the grid, hold the *Ctrl* key while dragging.

Rechaining lines

The best way to move a section of line (and the stations that are on it) to an existing trail is using the *Rechain* tool. To rechain lines in a project, right-click on the project in the layer view, and choose *Rechain Line* from the context menu.

To use the rechain tool, first click on a line to select it, as shown in Figure 74. This will actually only select the part of the line, beginning with the clicked point, which does not project onto itself.

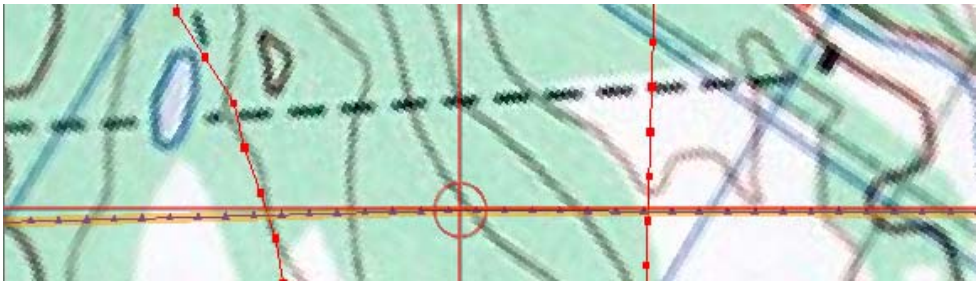


Figure 74 - Selecting a line segment

Next, hold the *Shift* key. You will notice that with the *Shift* key held down, the selection circle no longer appears when you bring the cursor close to a line. With the *Shift* key held, the rechain tool is ready for you to define a new segment of line. Since this segment might overlap other lines, selection must be disabled.

To define a new line segment, click on the first point of the segment without releasing the *Shift* key. A node will appear at the clicked point, as shown in Figure 75.

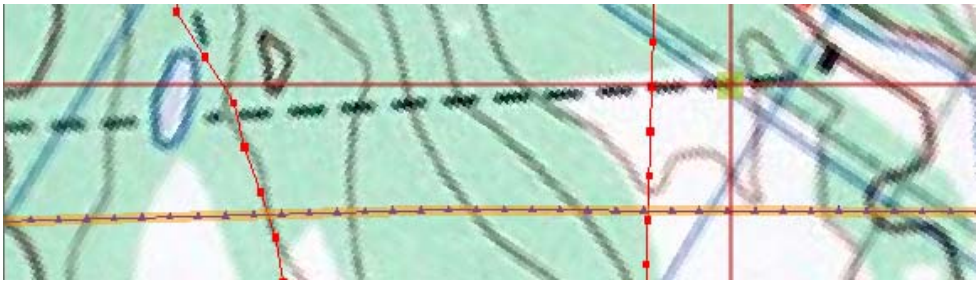


Figure 75 - Selecting the first point for rechain

Now click on the second point of the segment. As long as you hold the mouse button down, a line drawn between the first and second point will follow you, as shown in Figure 76, so you can align it with the existing trail. When you release the mouse button, a segment of the original line will be rechained to the new path, as shown in Figure 77.

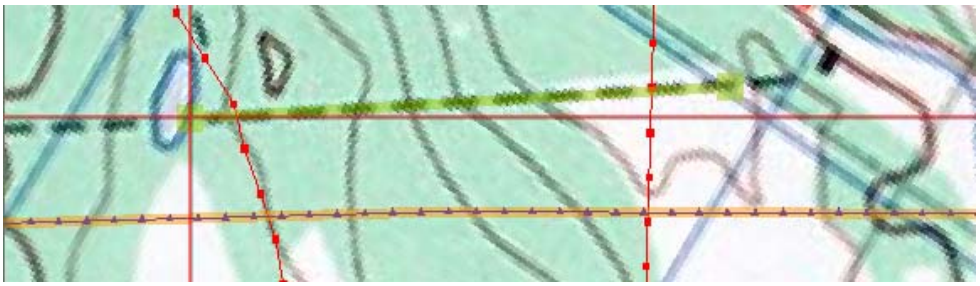


Figure 76 - Selecting the second point for rechain

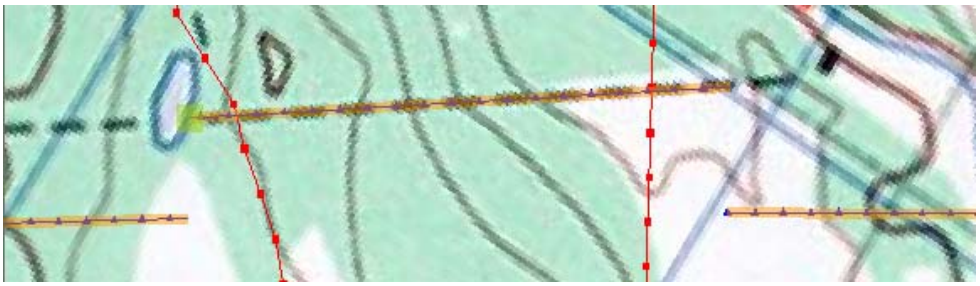


Figure 77 - Rechaining a line segment

Notice that a node remains at the last point clicked. By shift-clicking another new point, you can add another node connected to the new segment. Each shift-click adds a new segment. Every time you release the mouse button, a segment of the original line is rechained and the tool is ready for the next point to be entered. By shift-clicking a series of points, the line can be rechained to a complex path, as shown in Figure 78.

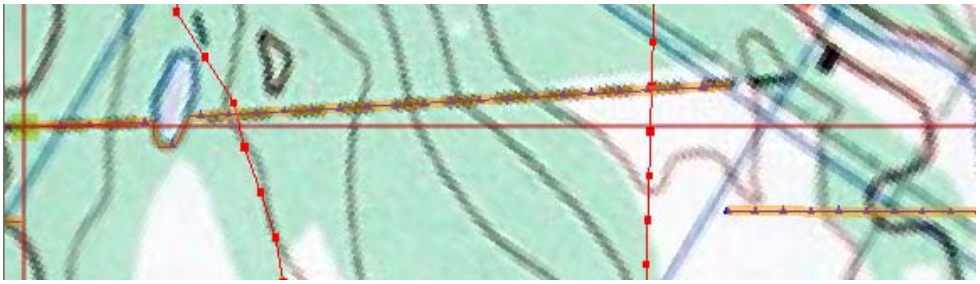


Figure 78 - Rechaining to a complex path

The *Clear* button in the tool can be used to clear the "active" point. This way, you can rechain multiple disjoint segments. To de-select the project line, release the *Shift* key and click in the background (when the selection circle is not visible).

Using the rechain tool, it is possible to rechain line segments rapidly by following the procedure above. This procedure is repeated here for clarity:

1. Click to select a line.
2. Holding the *Shift* key, define the new path.
3. Release the *Shift* key.
4. Repeat for other lines.

Snap to grid

See "Grids" on page 24 for more information.

Adding stations to a line

Stations can be added to a line either by interpolating between existing stations, or by adding stations to the end of the line. In either case, you will want to use the *Interpolate* tool. To interpolate stations in a project, right-click on the project in the layer view and choose *Interpolate Stations* from the context menu.

Interpolating stations

To interpolate between two existing stations using the *Interpolate* tool, click on the segment of line between the two stations, as shown in Figure 79.

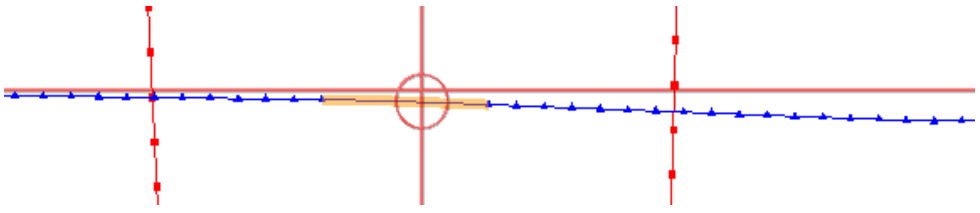


Figure 79 - Selecting a line segment for interpolation

With the segment selected, the interpolation tool (shown in Figure 80) will give details of the selected range, the measured distance, and number of missing stations.

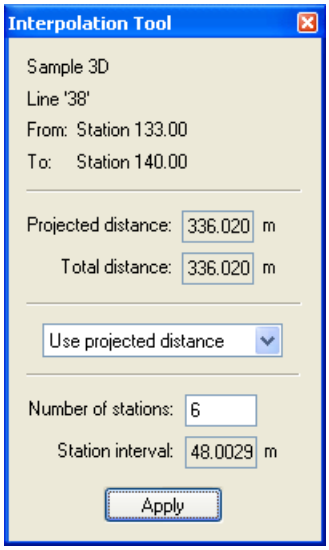


Figure 80 - The interpolation tool

By default, the interpolation tool will replace stations based on projected distance. It is also possible to interpolate along a path using the total distance by selecting "Use total distance" in the drop-down list. To interpolate a different number of stations from what is recommended, change the value labeled "Number of stations". Click *Apply* to interpolate the stations, as shown in Figure 81.

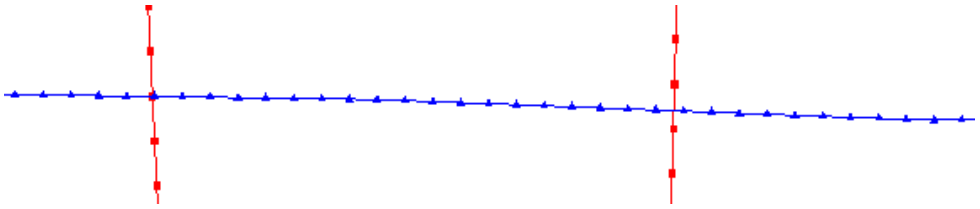


Figure 81 - After interpolating stations

Extrapolating stations

The procedure for extrapolating stations is similar to the procedure for interpolating them, except that instead of clicking on the line segment between two existing stations, you will click on the "tail" of the line which goes just past the last station, as shown in Figure 82. In this situation, the interpolation tool will extend the line segment as necessary to add the requested number of stations.

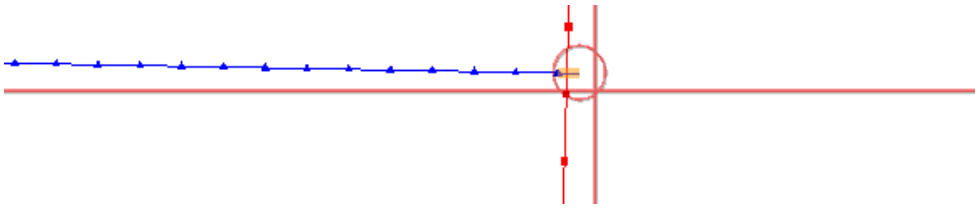


Figure 82 - Selecting the tail of a line for extrapolation

Adding a new line

To add a new line to a project, use the *Add New Line* tool. Too add new lines to a project, right-click on the project in the layer view and choose *Add New Line* from the context menu.

To begin using the tool, select a line in the map view by clicking on it. The tool options dialog will be populated with information about the line you've just clicked. Most of the values are self-explanatory. Station offset is the distance from the start of the line to the first station.

There are two ways to move ahead from here. If your line labels are integer values, you can press and release the *Shift* key to increment the line number by two. If you have selected a line other than the last one, you may need to press and release *Shift* several times. DirectAid will stop incrementing the line number when you have reached an unused number. This use of the *Shift* key should make more sense in a moment.

If your line labels are not integer values, or if you want to enter a specific line number, you can enter a new line label manually. The selected line will be de-selected in the map view, but the line parameters will remain unchanged. If you've entered the number of an existing line, that line will be highlighted and you will not be able to add a new line using that number.

Once you have entered a new line number, there should be no line selected in the map view. If you need to adjust any of the line parameters, this is the time.

Next, hold the *Shift* key. You will notice that with the *Shift* key held down, the selection circle no longer appears when you bring the cursor close to a line. With the *Shift* key held, the *Add New Line* tool is ready for you to define a new segment of line. Since this segment might overlap other lines, selection must be disabled.

To define a new line segment, click on the first point of the segment without releasing the *Shift* key. A node will appear at the clicked point, as shown in Figure 83.

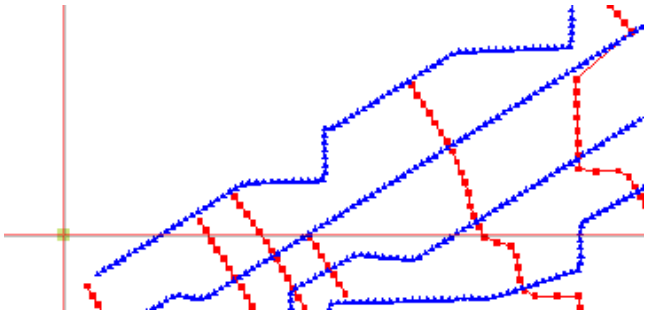


Figure 83 - Selecting the first point of a new line

Now click on the second point of the segment. As long as you hold the mouse button down, a line drawn between the first and second point will follow you, as shown in Figure 84, so you can align it with, for example, an existing trail. When you release the mouse button, a segment of the original line will be rechained to the new path, as shown in Figure 85.

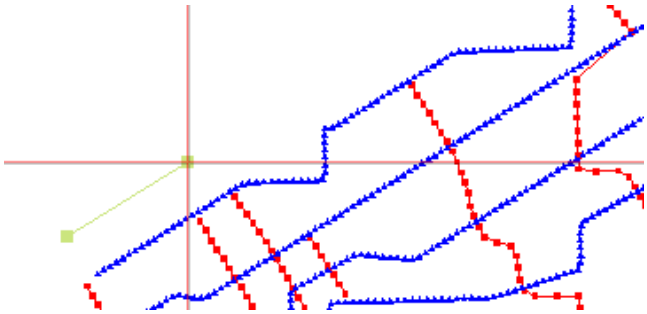


Figure 84 - Selecting the second point of a new line

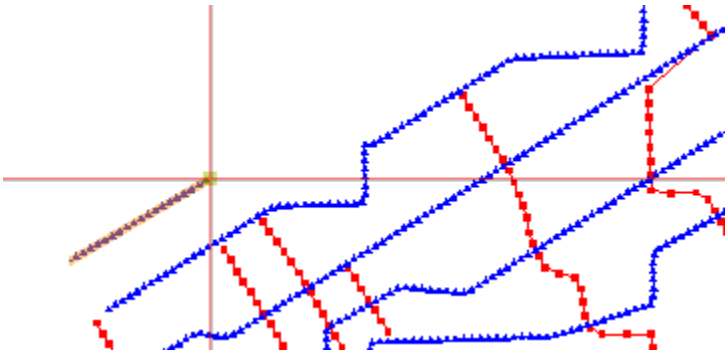


Figure 85 - Adding the first line segment of a new line

Notice that a node remains at the last point clicked. By shift-clicking another new point, you can add another node connected to the new line. Each shift-click adds a new segment. Every time you release the mouse button, a new segment of line is created and the tool is ready for the next point to be entered. By shift-clicking a series of points, a new line can be created which follows a complex path, as shown in **Error! Reference source not found.**.

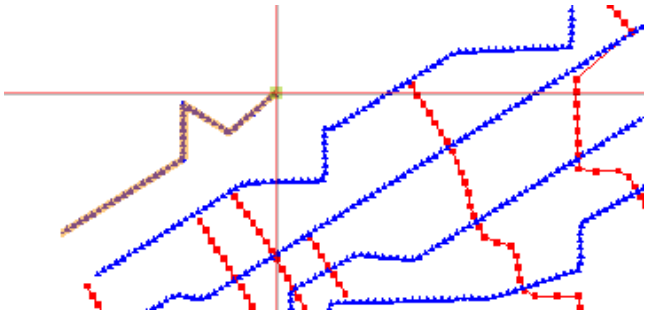


Figure 86 - Creating a new line following a complex path

One thing to note here is that stations are added to the line using *chained distance*. This is not in general the best way to add stations except for a straight line. Usually, if you need to add a line which follows a slightly perturbed path, you will want to add the line along the straight theoretical path, and then modify it using the *Rechain Line* tool described above. To add a new line along the theoretical path, you will need to use the snap-to-grid functionality described below.

Snap to grid

See “Grids” on page 24 for more information.

Deleting hangers

Suppose you have just trimmed a project to some geographic boundary, and this has left you with a lot of stations which don’t form complete “boxes” in the project, as shown in Figure 87.

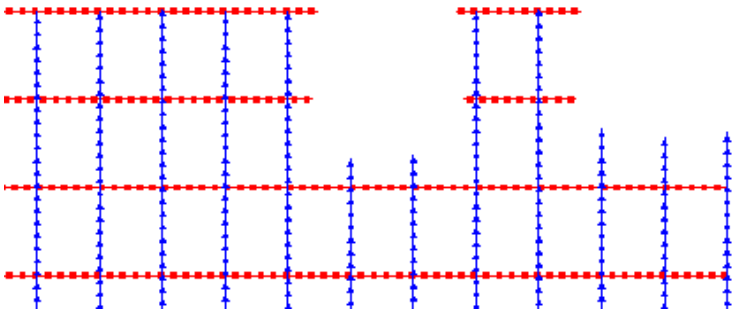


Figure 87 - A survey with "hangers"

It is possible to clean this up by selecting the individual salvos and deleting them. On a large survey, this could take a very long time. For that reason, we have developed the *Delete Hangers* tool. To access this tool, right-click on a project layer in the layer view and choose *Delete Hangers* from the context menu. The *Delete Hangers* tool is shown in Figure 88.

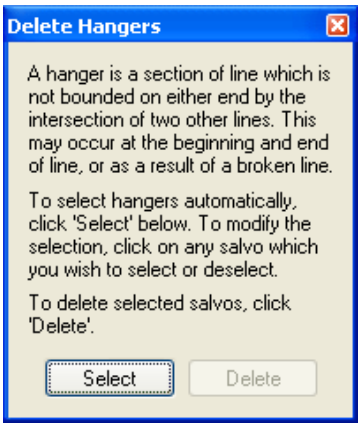


Figure 88 - The Delete Hangers tool

To use the tool, click *Select*. The tool will highlight all the hangers in the tool colour (by default, green), as shown in Figure 89.

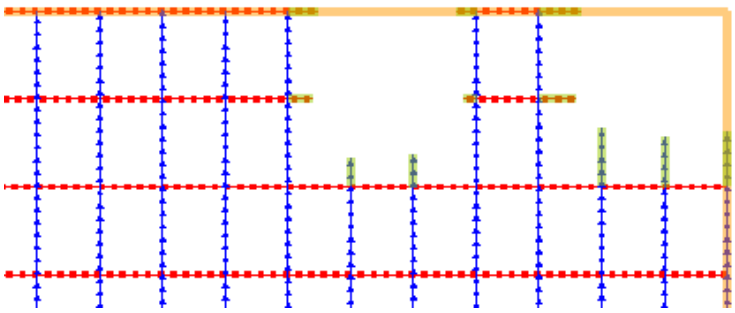


Figure 89 - The Delete Hangers preview

The highlighted segments, called “hangers”, are really just sections of line which are not bounded on both sides by an intersection with another line. In some cases, particularly when you have imported the survey from an external source, the tool may identify several incorrect hangers because a line doesn’t quite make it to the intersection. Individual sections of line can be toggled as marked/unmarked by clicking on them.

When you have checked that all the hangers are correctly identified, click *Apply*. The tool will delete the marked sections, as shown in Figure 90.

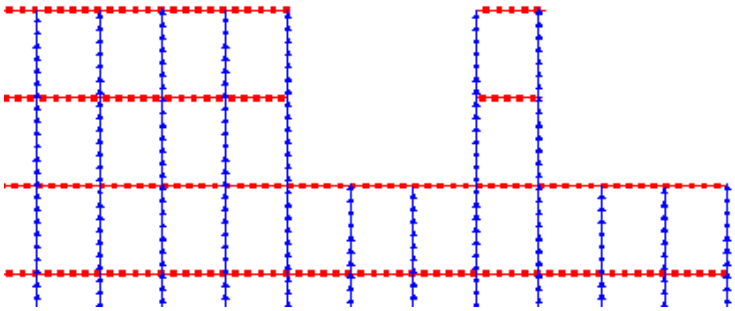


Figure 90 - The same survey without "hangers"

Rotating and moving a project

To rotate or move a project, use the *Rotate/Move Project* tool. To use this tool, right-click on a project in the layer view and choose *Move Project* from the context menu. The tool is shown in Figure 91.

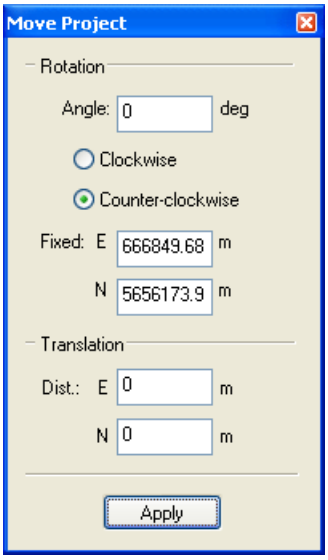


Figure 91 - The Rotate/Move Project tool

This tool can be used to rotate or move all or part of a project. Only visible portions of the project will be moved, so it's possible to move, for example, just the receivers by un-checking the sources in the *Layer View*.

Changes can be made manually or by dragging in the map view. To rotate or move the project manually, first select a project from the drop-down list at the top of the tool. Next, enter the rotation angle and direction, fixed point location, and translation distance. Check your values and click *Apply*. To apply a rotation only, set the translation distance to zero. To apply a translation only, set the rotation angle to zero.

To move the project in the map view, click and drag the project in the map. A “ghost” of the project will follow, as shown in Figure 92. When you release the left button, the entire project will move into place. For very large projects, this may take a moment.

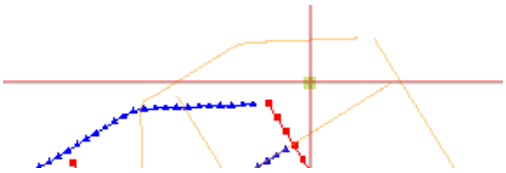


Figure 92 - Dragging a project in the map view

To rotate a project in the map view, first click to define the fixed point. This is the center of the rotation, and is marked with a green dot in the map. Next, hold *Shift* and click and drag in the map. Once again, a “ghost” of the project will follow. The further away from the fixed point you click, the better, since you are defining an “arm” which will control the rotation. If you click too close to the fixed point, the rotation may appear to be inaccurate.

These two actions can be used, for example, to change a project’s northeast corner and line it up with existing trail very quickly. First, click and drag the northeast corner where you would like it to go. This has the side-effect of setting the fixed point. Next, move to the other end of the line. While holding *Shift*, click and drag the other end of the line so that it also lies along the existing trail. The project is now in place.

Merging projects

You will sometimes need to take two projects and make them one. One possibility is that you’ve generated theoretical positions for all your stations, and are now receiving surveyed positions. To handle this, load the theoretical stations into one project and the surveyed stations into another, as shown in Figure 93.

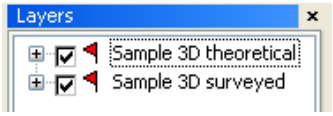


Figure 93 - Two projects loaded

Now drag the “surveyed” project onto the “theoretical” project. The “theoretical” project should be highlighted just before you release the left mouse button.

When you release the button, you will be asked to confirm the merge operation. Click *OK* to continue. If the two projects have different coordinate systems, copied lines and stations will be converted to the target’s coordinate system. The merge process will splice sections of line (along with their stations) from the source project into the target project. Where possible, the boundaries of the splice will be joined.

Renumbering lines

To renumber all the receiver or source lines, expand the project layer in the layer view and right-click on “Receivers” or “Sources”. From the menu, choose *Renumber Lines*. It is also possible to renumber just a few lines by selecting them in the layer view and right-clicking on the selection. From the menu, choose *Renumber Lines*. This will bring up the dialog box shown in Figure 94.

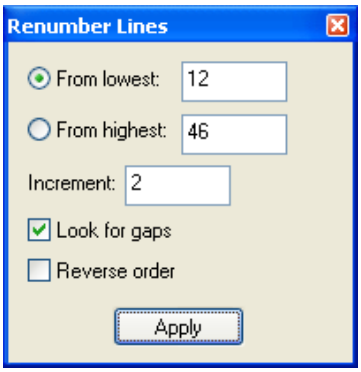


Figure 94 - Renumbering lines

Our goal with the *Renumber Lines* tool was to design a tool which makes simple renumbering tasks easy, and hard ones possible. For simple renumbering tasks, you will usually renumber all the receiver lines or all the source lines. The tool will automatically detect the lowest and highest lines in the selection, as well as the line number increment.

By default, the tool will look for missing lines, and adjust the line numbers accordingly. To disable this feature, and renumber lines in sequence regardless of the space between them, uncheck the *Look for gaps* checkbox.

Also by default, the tool will retain the current order of the lines. If your line numbers increase from north to south before the operation, they will do so afterward as well. To reverse the order of the lines, check the *Reverse order* checkbox.

Finally, you can choose to renumber the lines starting with the highest number, or with the lowest. If your line numbers increase from north to south, for example, then renumbering from the lowest line number will fix the number at the north end of the survey, and increment going south. There is no need to make the increment negative if you are renumbering from the highest line number—DirectAid will automatically decrement instead of incrementing. Once you have set everything up, click *Apply* to renumber the selected lines.

In the extreme, the *Renumber Lines* tool can be used to renumber individual lines. Just right-click on the line in the layer view and choose *Renumber Lines* from the menu. In this case, it doesn’t matter if you renumber from lowest or highest, look for gaps, or reverse order. To renumber a single line, just enter the new line number in the *From lowest* edit box, and click *Apply*.

If you have specified a renumbering scheme which results in two lines having the same line number, DirectAid will warn you that the operation cannot be completed. Lines must have unique line numbers.

Renumbering stations

The *Renumber Stations* tool can be used to renumber all the sources or receivers, just the stations in a single line, or any group of stations that you can select. To renumber all the sources or receivers, expand the project layer in the layer view and right-click on "Receivers" or "Sources". From the menu, choose *Renumber Stations*. To renumber just the stations in a single line, right-click on the line and choose *Renumber Stations*. To renumber a selection of stations, first use the *Select Objects* and *Select Area* tools to make your selection. Right-click on the selection and choose *Renumber Stations* from the context menu. Any of these methods will bring up the dialog box shown in Figure 95.

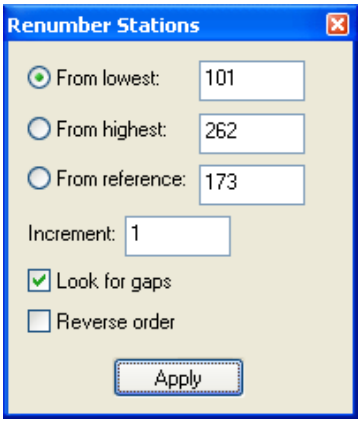


Figure 95 - Renumbering stations

The *Renumber Stations* tool options are very similar to the options for *Renumber Lines*, described above. The tool automatically detects the lowest and highest station numbers in the selection. If you renumber from the lowest station number, for example, then the first selected station on each line will be given that number.

Often, you want to renumber the stations on each line so that they "line up" along some axis. For this reason, we have introduced the *renumber from reference* option. If you select this option, a green line will appear in the middle of the survey. You can click and drag anywhere in the map view to move the line. When you renumber using this method, DirectAid will begin numbering where the reference line intersects the project line, as shown in Figure 96.

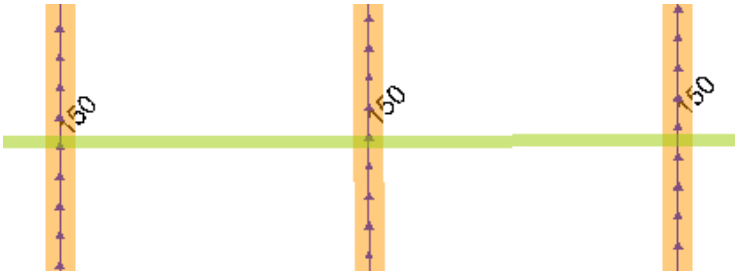


Figure 96 - Renumbering stations using a reference line

There are some subtleties to using the reference line. DirectAid renumbers first in the forward direction, starting at the reference line. If you have enabled the *look for gaps* option, then it will look for a station within half a station interval of the reference line. If it does not find a station, it will "skip" station numbers as needed. When it is done renumbering in the forward direction, DirectAid renumbers in the reverse direction, beginning with the first station it found in the forward direction. In Figure 96, we have renumbered from the reference line with a station number of 150. As you can see, for the line on the left this has actually placed 150 just below the line, since the station in the forward (upward) direction is more than half a station interval from the reference line.

Stations are normally renumbered with station number increasing from the beginning of the line to the end of the line. It is possible to reverse the numbering for complete lines by checking the *Reverse order* checkbox. The checkbox will only be enabled if you have selected complete lines. This is because station numbers must always increase going from the beginning to the end of the line, so it is not possible to reverse the numbering for a portion of the line.

Once you have set everything up, click *Apply* to renumber the selected stations. If you have specified a renumbering scheme which results in two or more stations having the same station number, DirectAid will warn you that the operation cannot be completed. Stations must have unique station numbers.

The measuring tool

The measuring tool can be used to measure areas and lengths in the map view. To use the measuring tool, either select *Measure* from the *Tools* menu, or click the button in the toolbar ( ). This will bring up the dialog box pictured in Figure 97.

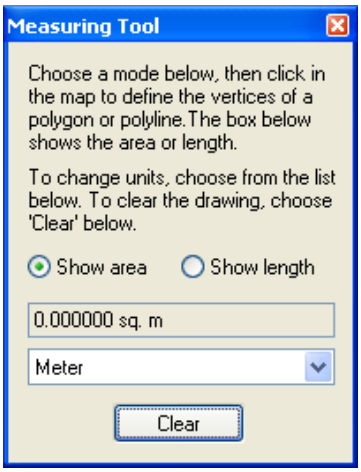


Figure 97 - The measuring tool

To use the measuring tool, first choose either *Show area* or *Show length*. Then click in the map to define vertices. If you are measuring an area, the tool will draw the area as you define it, as shown in Figure 98.

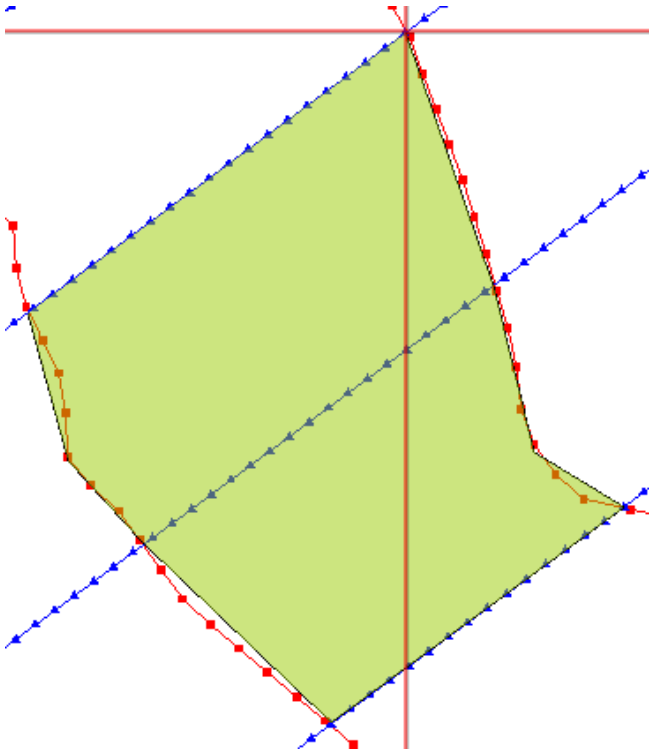


Figure 98 - Measuring an area

If you are measuring a length, the tool will draw the edges as they are defined, as shown in Figure 99. To change measurement units, choose from the list at the bottom of the dialog box. To clear the vertices, click the *Clear* button.

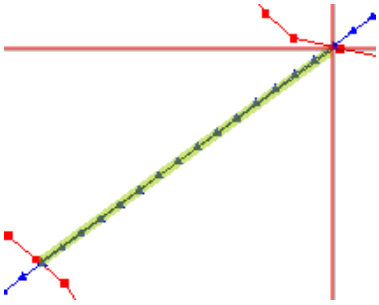


Figure 99 - Measuring a length

Breaking and joining lines

The *Break/Join Lines* tool can be used to break one line into two, or join two lines to make a single line. To use the tool, right-click on a project in the layer view and choose *Break/Join Lines* from the context menu.

To break a line between two stations, click between the stations in the map view, as shown in Figure 100.

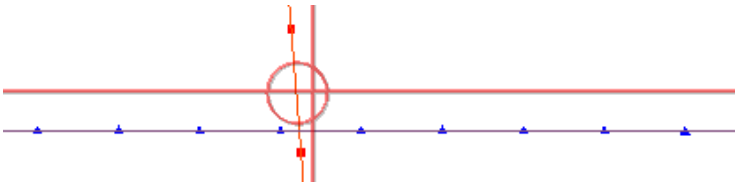


Figure 100 - Breaking a line

When you click on the line, DirectAid will ask you to provide a new line label for the shorter of the two newly created line halves. The line label must be unique. Click *OK* to finish breaking the line.

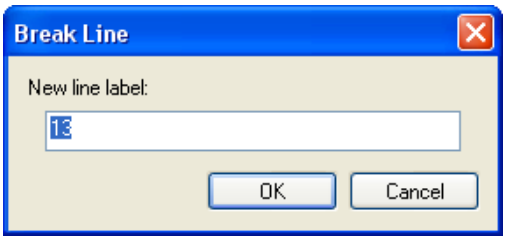


Figure 101 - Choosing a new line label

A newly broken line is shown in Figure 102. Note that the line break is automatically moved to the nearest salvo edge, if possible.

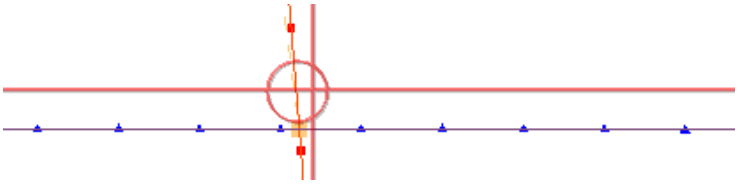


Figure 102 - The result of breaking a line

Before joining two lines, you must make sure that the station numbering of the two lines does not overlap, and that the joined line will be numbered sensibly. For example, if you want to join a small stub line onto the end of a longer line, then the stub line's numbering must begin after the last station of the longer line. To renumber the stations in one of the lines, use the *Renumber Stations* tool described on page 63.

To join two lines, click and drag from the EOL of one to the BOL of the other, or vice versa. Allowable targets are drawn with a small symbol, as shown in Figure 103. The cursor will snap to them.

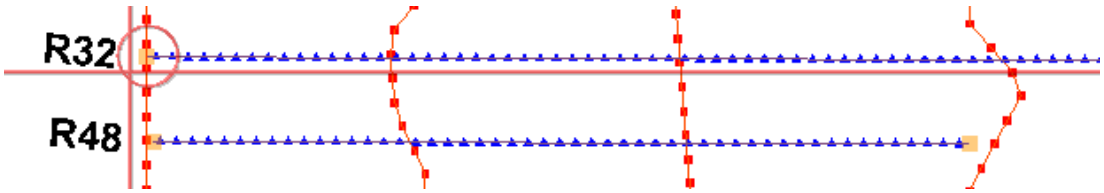


Figure 103 - Line end targets

When you click on one line end, only the line ends it can be connected to will remain highlighted, as shown in Figure 104.

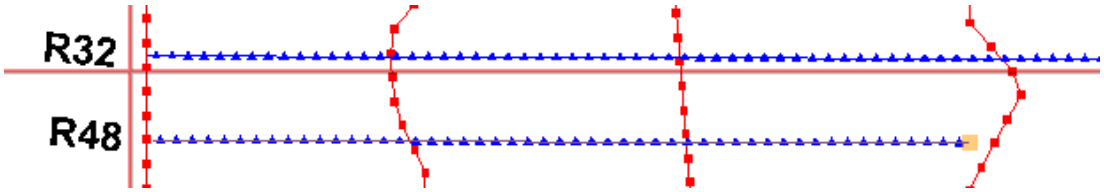


Figure 104 - Line end targets after picking first end

Dragging to one of the allowable targets will draw a temporary line between the two ends, as shown in Figure 105.

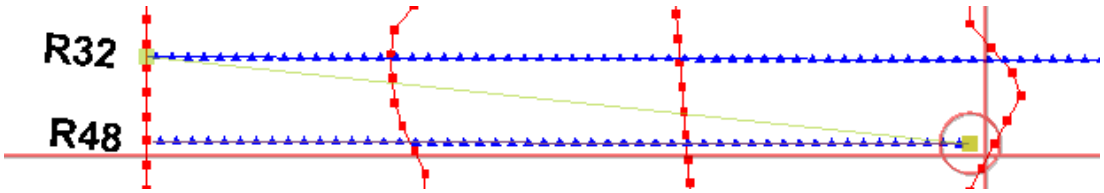


Figure 105 - Ready to connect two lines

Finally, letting go of the mouse button will complete the operation, and cause the two lines to be joined, as shown in Figure 106. The new line will share the same name as the longer of the two original lines.

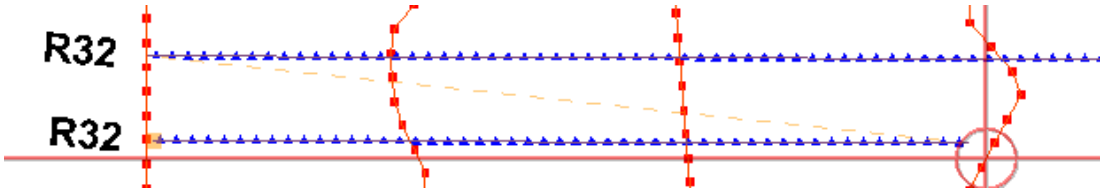


Figure 106 - Two lines connected

Locking a project

The geometry of a locked project—the position of its lines and stations—cannot be changed. To lock or unlock a project, right-click on the project in the layer view and choose *Lock* from the context menu. When a project is locked, the *Lock* menu item will be checked, and a small "lock" symbol will also appear in the layer view, as shown in Figure 107.

Figure 107 - A locked layer

Observer's Logs

Importing observer's logs

The observer's log import can read either ARAM text files, or the full Sercel observer's log.

To import an observer's log, right-click on a project in the layer view and choose Import and then Observer's logs from the context menu. In the Open dialog, pick a type using the Files of type dropdown at the bottom. By default, the dialog looks for files ending in ".txt". If you want to find a file which doesn't end in ".txt", just enter "*.*" in the File name, hit enter, and the file list should be updated to include everything. Choose the file to import and click Open. For each source station listed in the log, DirectAid will update the "_Date_Shot" property to the date indicated in the log.

Marking shots

Now that the properties have been imported, you probably want to do something with them. The first thing to do is probably to see what's been shot. To do this, go to *Manage Fields* in the *Tools* menu. Select the "_Date_Shot" property, and click *Options*. Click *Visualization*. From the drop-down, choose *Points*. This will mark every station with the

"_Date_Shot" property as indicated by the rules below. The simplest thing is just to mark all the stations which have been shot. To do that, use settings such as those shown in Figure 108.

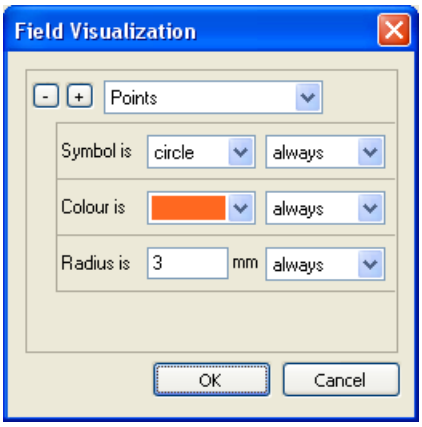


Figure 108 - Marking shot sources

This will mark each station which has been shot with a 3 mm radius orange circle. If we just want to mark the ones which were shot on a particular date, we can do this as shown in Figure 109.

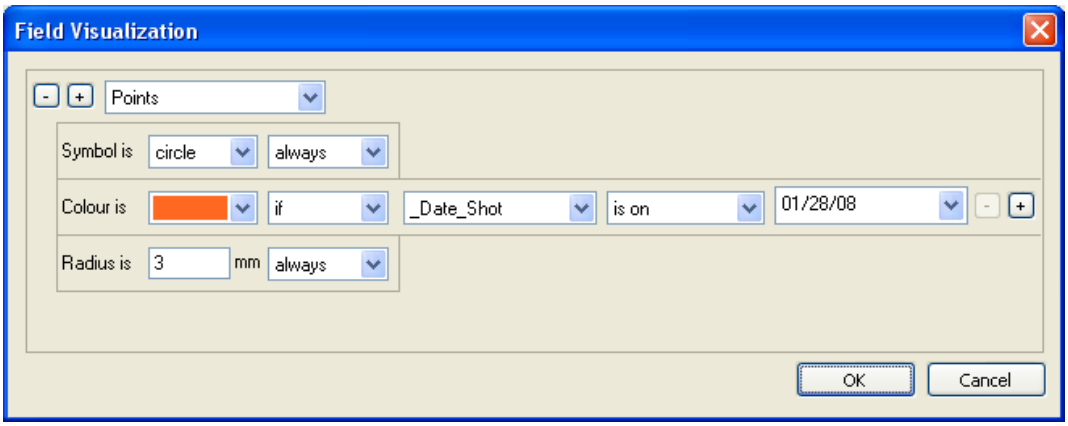


Figure 109 - Marking sources shot on a particular date

This will mark every station shot on January 28, 2008 with a 3 mm radius orange circle. Finally, we can chain several of these rules together by clicking the "+" sign to the right of the colour. This will add a second line, which will be used if the first one fails. For example, we might use the chain of filters shown in Figure 110.

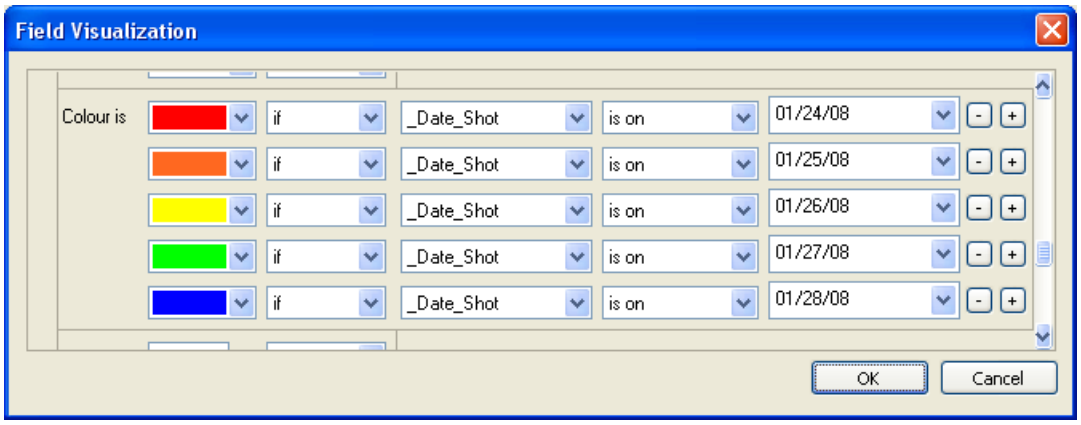


Figure 110 - Marking sources shot on a range of dates

This would mark stations with different colours depending on the date they were shot. If we wanted to handle wider ranges of dates, we adjust the filters as shown in Figure 111.

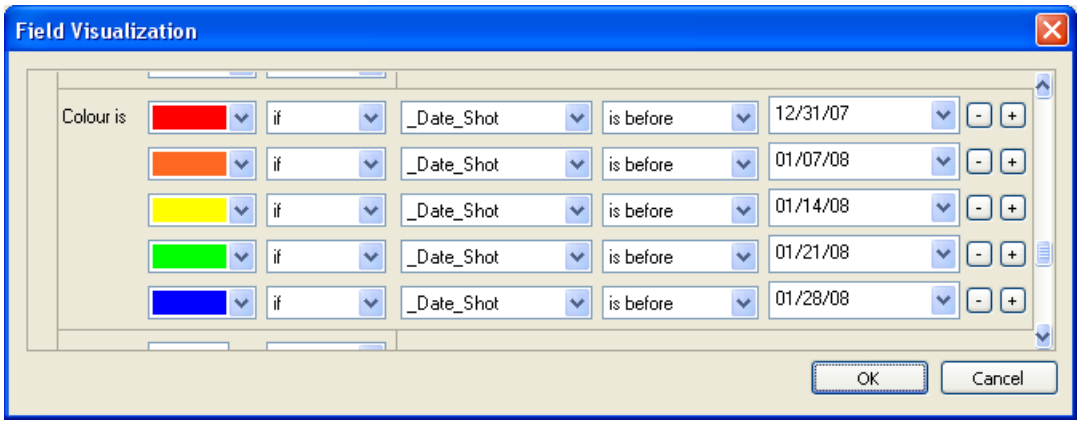


Figure 111 - Marking sources shot on a wide range of dates

Because the filters act in the order they're entered, this will colour stations according to the week they were shot.

What has not been shot?

Finally, you can quickly generate a report indicating which sources have not been shot. Change to the "Reports" tab and open the "Miscellaneous reports" folder. Right-click on "Unshot sources" and choose *Run* from the context menu. The first time you do this, it may take a moment to compile the Python scripts. You will be presented with report options which affect the appearance of the report. Click *OK* to continue, and the report should open in Adobe Reader. Ranges of unshot sources are itemised by line. If an entire line is unshot, the right column will say "all". If every station is shot, the column will read "none".

Drawings

Reading/writing drawings

Reading

To import vector data, choose *Vector layer* in the *Import* pop-up menu. You will first be asked to select the files to import. For vector layers, files imported in a single operation are loaded into separate layers, and may have different coordinate systems. When you have selected one or more files, click *Open* to continue.

Next, the vector import wizard will appear. This wizard contains the three coordinate system pages. When you have entered any required information in the import wizard, click *Finish* to import the first vector file. When the import is complete, the process will be repeated for any other vector data you may have selected for import. As usual, you may need to un-zoom () in order to see the drawings.

Writing

DirectAid can export a drawing layer as a DXF or ESRI Shapefile in any coordinate system. Thus, it can be used to convert between DXF and Shapefile formats, and between coordinate systems as well. To export a drawing layer, right-click on the layer in the layer view and choose *Export* from the context menu. After giving the file a name, you will see the three standard coordinate system pages.

Lastly, you will be asked to specify a “precision” for the exported file. Because a straight line in one coordinate system may be a curve in another, it is sometimes necessary to represent a single edge as several edges when converting a drawing. The precision parameter determines when this kind of edge division should end. Lines in the exported file will not err by more than the specified distance. Verify the export coordinate system and precision, then click *Finish* to complete the operation.

DirectAid can also export a project layer as a DXF or ESRI shapefile. To export a project layer as a drawing, right-click either the project layer or the “Source” or “Receiver” layer in the project, and choose *Export* from the context menu. In the drop-down list labeled “Save as type” at the bottom of the *Save As* dialog, you should see both DXF and Shapefile as options. Choose one of these two file types and click *Save*. Choose a coordinate system and precision as above, then click *Finish* to complete the operation.

Editing drawing layers

Most of the properties of a vector object are accessible either through the *Properties* dialog box, or through map object selection mode in the map view. A vector layer can contain five basic objects: groups, points, polylines, polygons and text.

Groups

A *group* collects several vector objects into a single parent object. A group may contain any of the five basic objects, including sub-groups. When a group is in a *grouped* state, actions on the group, for example selection and dragging operations, will act on every member of the group. If a group is temporarily *ungrouped*, it acts as a transparent folder for its members, which can be selected and edited independently.

To *ungroup* a group temporarily, first select the group. A selected group is drawn with a *bounding box* surrounding the selected group members, as shown in Figure 112.

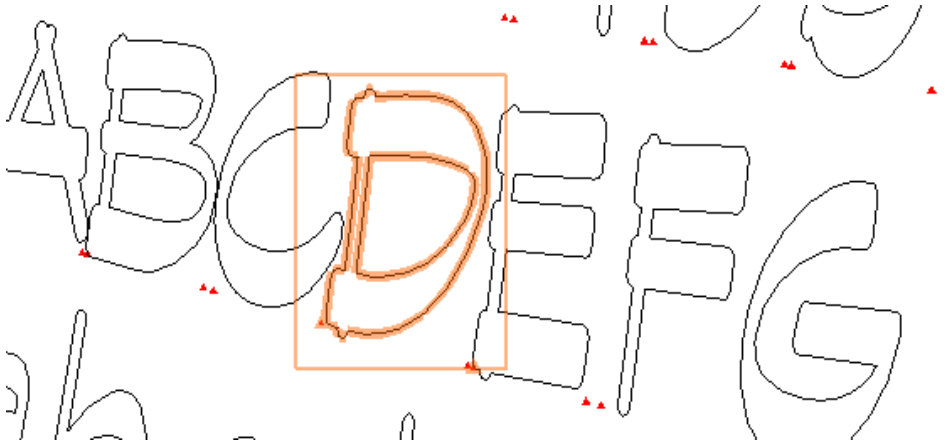


Figure 112 - A selected group

With the group selected, double-click on any selected group member to ungroup it temporarily. Every member of the group will remain selected, but the bounding box will not be drawn. A temporarily ungrouped group is shown in Figure 113.

A temporarily ungrouped group will return to its grouped state when none of its children are selected. Otherwise, it will remain ungrouped. This means that we can, for example, edit the nodes of a child polygon by double-clicking on the ungrouped child.

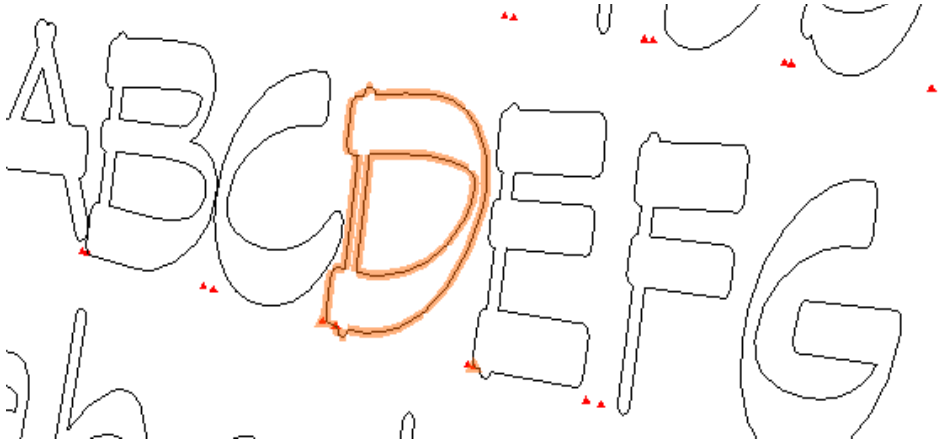


Figure 113 - An ungrouped group

Points

A *point* is the simplest vector object. It is a position marked with some symbol. In DirectAid, a station is implemented as a special case of a point.

The *Properties* dialog for a point, or for any layer containing a point, has a *Point Style* page, shown in Figure 114. Note that the symbol size is specified in page millimetres.

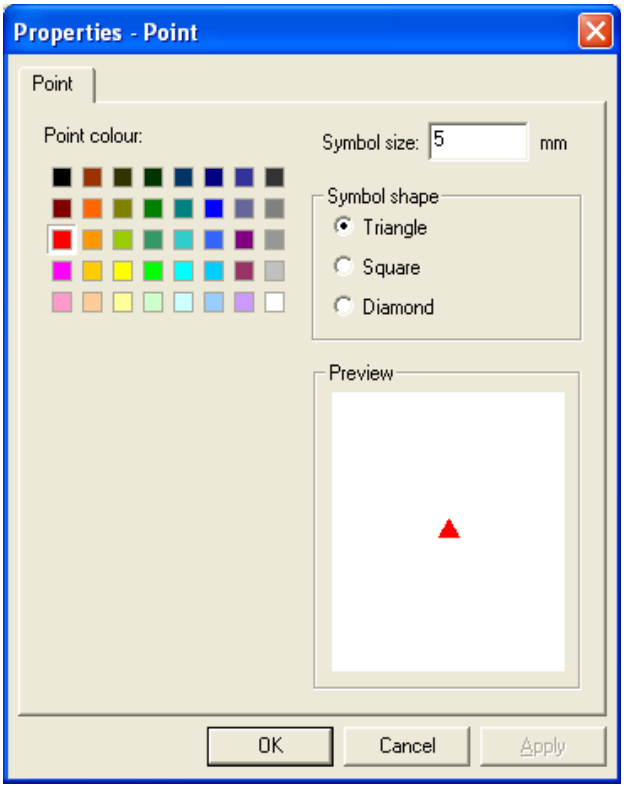


Figure 114 – The point style page

In addition to its properties, there is a single operation which can be performed on any point in the map view. To move a point, just select the point and drag it as you would a station.

Polylines and polygons

A *polyline* is a series of points connected by edges. Each edge may be connected, in which case it is normally displayed, or it may be broken, in which case it is displayed only as a thin line in node editing. A *polygon* is simply a polyline whose first and last points are defined to be the same. In DirectAid, a project line is implemented as a special case of a polyline.

The *Properties* dialog for a polyline, or any layer containing a polyline, has a *Line Style* page, shown in Figure 115. Once again, note that line weight is specified in page millimeters.

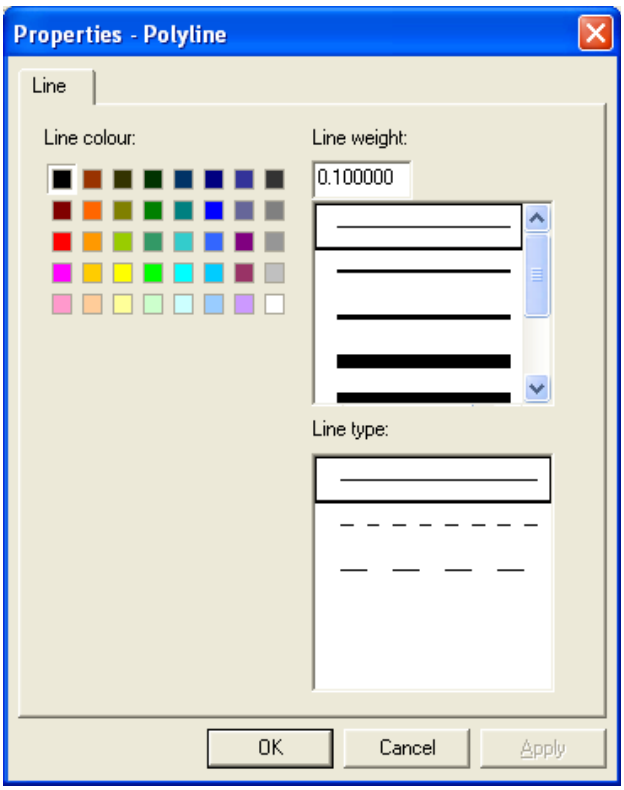


Figure 115 - The line style page

To edit a polyline, select the polyline then double-click the selection. The details of polyline editing are given in the section ”Editing lines” on page 51.

Text

A *text box* in DirectAid is defined in source coordinates. That is, the box and the text inside it lie at a particular position on the ground, regardless of the camera coordinate system. The *Properties* dialog for a text box, or any layer containing a text box, has a *Text Style* page, which is divided into five stacked sub-pages: alignment, font, layout, text, and text box. To switch between the sub-pages, use the drop-down box located in the top-right corner of the *Text Style* page. At the bottom of the page is a preview of the text within the box.

The text box is positioned in source coordinates relative to an *alignment point*. The *Alignment* sub-page, shown in Figure 116, specifies how the text box will be positioned relative to the alignment point. Horizontally, the alignment point may lie at the left edge of the box, in the center, or at the right edge of the box. Vertically, the alignment point may lie at the bottom edge of the box, in the middle or at the top edge of the box. If the text box contains a single line of text, the alignment point may lie along the baseline of the text.

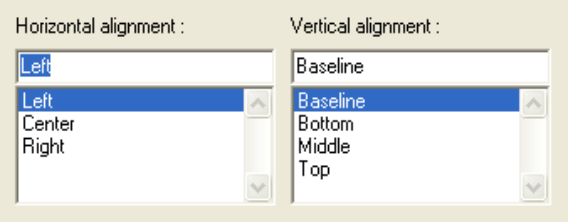


Figure 116 - Text alignment properties

The *Font* sub-page, shown in Figure 117, specifies the font name, size, style and colour which will be used for text within the box. Note that the font size is measured in source units and specifies a size *on the ground*.

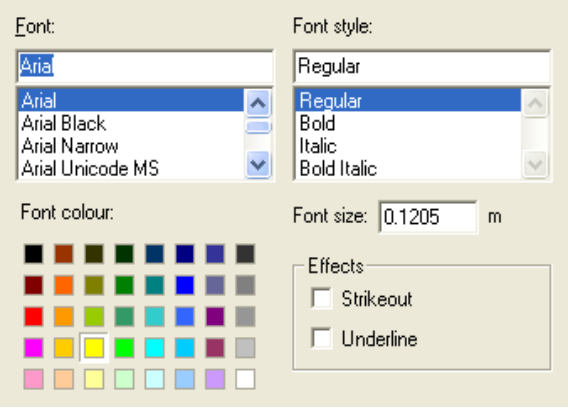


Figure 117 - Text font properties

The *Layout* sub-page, shown in Figure 118, specifies how the text will be arranged within the text box. Line spacing is the distance between the baselines of consecutive lines of

text, measured in multiples of the height of a single line of text. In addition, the *Layout* sub-page contains options to mirror the text horizontally or vertically, and to enable to disable word wrap.

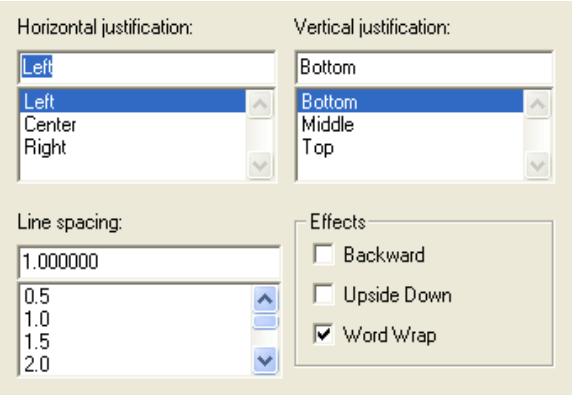


Figure 118 - Text layout properties

The *Text* sub-page contains the actual text which will be drawn in the text box. Use this page to modify the text.

Finally, the *Text Box* sub-page, shown in Figure 119, specifies the text box dimensions and background colour. To the left of each dimension is a check box labeled *Auto*. When the box is checked, that dimension will be automatically calculated based on the dimension of the enclosed text and margins. When the box is unchecked, the dimension is fixed to a user-specified value. Note that these dimensions are measured in source units and specify a size *on the ground*.

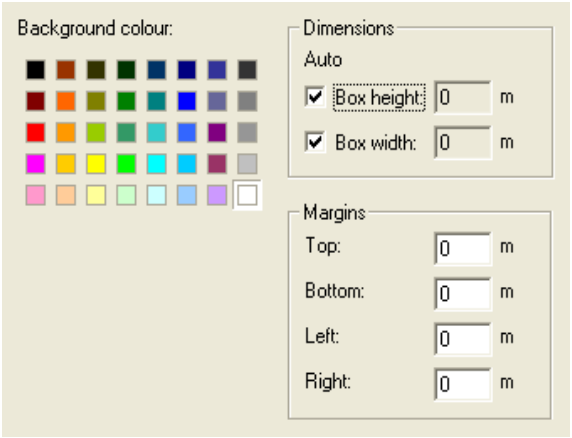


Figure 119 - Text box properties

In addition to these properties, a text box can be edited in the map view. To edit a text box, change to map object selection mode and select a text box. The text box will appear with a transparent overlay, as shown in Figure 120.

Figure 120 - Text box selection

From this mode, a text box can be dragged. To edit the box orientation and size, double-click the selected text box. When you are in editing mode, the text box will appear with a transparent overlay, as shown in Figure 121.

Figure 121 - Text box editing

To move the whole text box, drag the node located at the centre of the box. To change the dimensions of the box, drag the nodes located along the boundary. Note that if your text box is centre-aligned either horizontally or vertically, the box will grow symmetrically about that axis. To change the orientation of the text box, drag the point halfway between the centre and the right edge.

Adding to a drawing

To add to a drawing layer, right-click on it in the layer view and choose *Draw* from the context menu. This will bring up the *Vector Draw Tool*, which can be used to draw new objects in an existing vector layer.

To create a new vector layer, right-click on the white space in the layer view and choose *New* then *Vector Layer*.

Points

To add points to a vector layer, choose the leftmost button in the *Vector Draw Tool*. The tool is shown in point mode in Figure 122. In the bottom of the tool, choose the colour, size, and symbol for the points, then click in the map view to define new points.

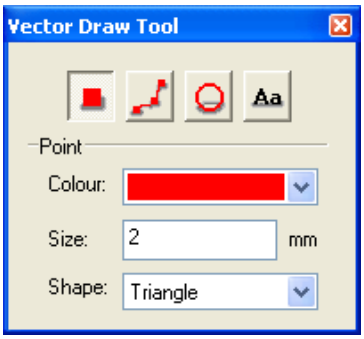


Figure 122 - Drawing points

To edit or delete existing points, you will need to exit the *Vector Draw Tool* and follow the instructions above for point editing.

Polylines

To add polylines to a vector layer, choose the second button in the *Vector Draw Tool*. The tool is shown in polyline mode in Figure 123. In the bottom of the tool, choose the colour, weight, and pattern for the line, then click in the map view to define new vertices. Double-click on the last point to finish a polyline.

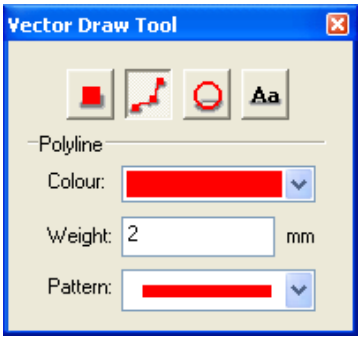


Figure 123 - Drawing polylines

Circles

To add circles to a vector layer, choose the third button in the *Vector Draw Tool*. The tool is shown in circle mode in Figure 124. In the bottom of the tool, choose the colour, line weight, line pattern, and optionally the radius for the circle. To draw a circle with a predefined radius, set the radius in the tool and then click in the map to define the center of the circle. A circle can also be drawn by clicking and dragging in the map, to define first the center of the circle and then the radius. Subsequently clicking in the map will draw more circles with the same radius.

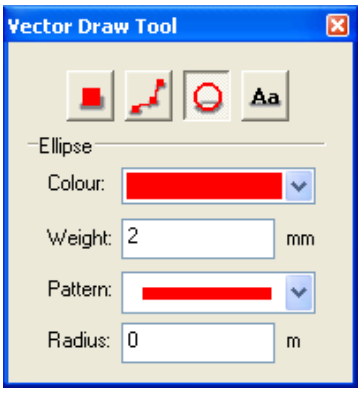


Figure 124 - Drawing circles

Text

To add a text label to a vector layer, choose the rightmost button in the *Vector Draw Tool*. The tool is shown in text mode in Figure 125. In the bottom of the tool, choose the colour, font, style, size, angle, and contents for the text box. To draw a text label with a predefined height and orientation, specify the size and angle in the tool and then click in the map to define the left edge of the text box. Alternatively, you can click and drag in the map to define first the left edge, and then the size and orientation of the text box. Subsequently clicking in the map will draw more labels with the same size and orientation.

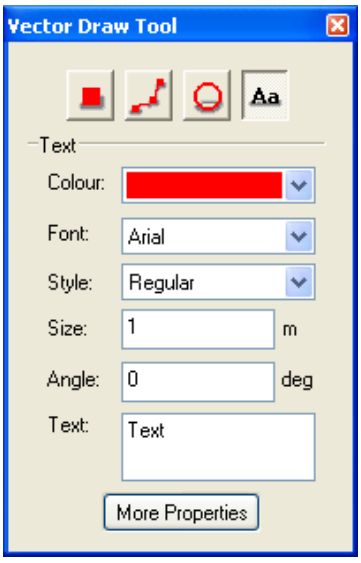


Figure 125 - Drawing text

Images

Reading images

To import an image, choose *Image layer* in the *Import* pop-up menu. You will first be asked to select the files to import. For image layers, files imported in a single operation are loaded into separate layers, and may have different coordinate systems. When you have selected one or more files, click *Open* to continue.

What happens next depends on the auxiliary information available for the image. This information may be in the form of an OziExplorer MAP file or a TFW file. If you are importing a TIFF file, this information may also come from GeoTIFF tags. Information will be read automatically from any auxiliary file which has the same filename but a different extension. DirectAid 2.0 first looks for a MAP file, then a TFW file, and finally it looks for GeoTIFF tags if they are available. If it cannot find any of these data sources, or if the information in the files is incomplete, DirectAid will ask which auxiliary file you would like to use. This dialog is shown in Figure 126.

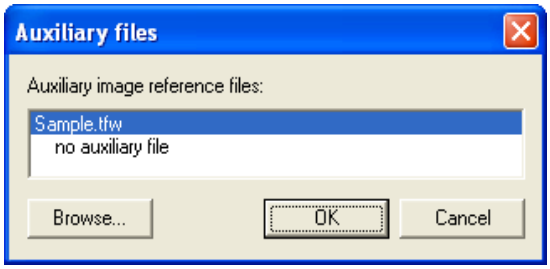


Figure 126 - Auxiliary files

To use the information from a particular file, choose that file and click *OK*. To browse for an auxiliary file which may be in another location, or may not have a filename similar to the image file, click *Browse*. If you wish to enter georeferencing information for the image manually, choose “no auxiliary file” and click *OK*. For more information on rectifying an image manually, see “Image rectification” below.

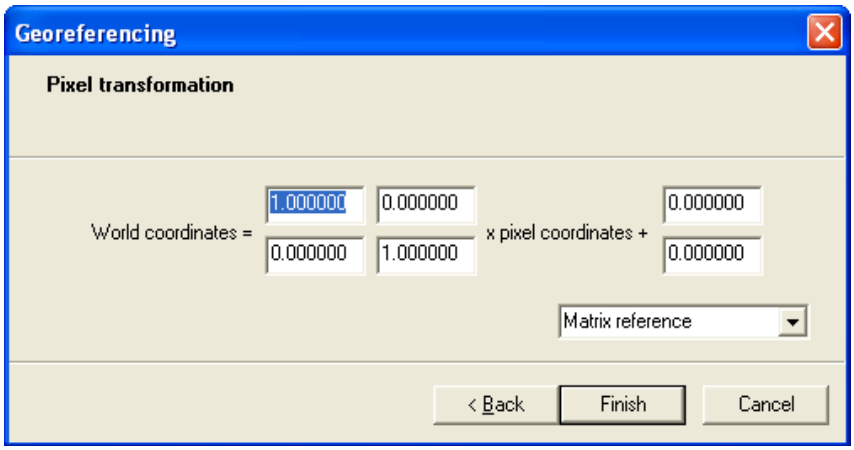


Figure 127 - Image georeferencing

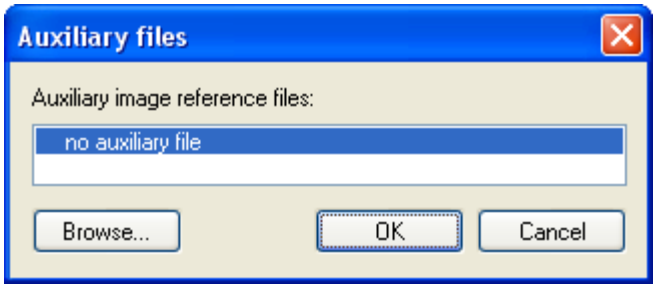
If the auxiliary information for the image was incomplete, the image import wizard will appear. Only information which has not been specified in auxiliary files will be requested at this point. The image import wizard may include the three coordinate system pages. It may also include a georeferencing page, as shown in Figure 127. This page allows the user to edit the matrix used to transform pixel coordinates to source coordinates. By choosing other options from the drop-down box in the bottom right of the page, it is also possible to reference the image by choosing pairs of pixel and source coordinates. However, at this time only matrix referencing can be used on import.

When you have entered any required information in the import wizard, click *Finish* to import the first image file. When the import is complete, the process will be repeated for any other images you may have selected for import. As usual, you may need to un-zoom ( ) in order to see the images. If you still cannot see the images, you may have chosen a datum or projection which is not valid for the project area. Another possibility is that you are looking at the back side of the image, which is not visible. If your camera projection is set to ECEF, you may need to rotate the globe to make your image visible. Otherwise, you should be able to correct the problem by re-squaring your camera axes ( ) and un-zooming again.

Image rectification

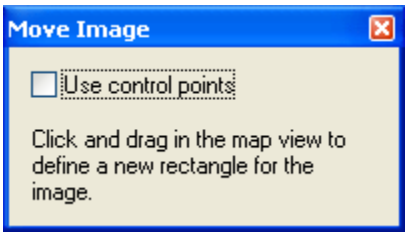
Using DirectAid, you can rectify images which come without georeferencing information. You will still need to know what coordinate system the image belongs in, but we have made it easy to position images relative to other layers.

To read an unreferenced image, import the image as usual. When you are asked to identify an auxiliary file, select “no auxiliary file”, as shown below, then click “OK”.



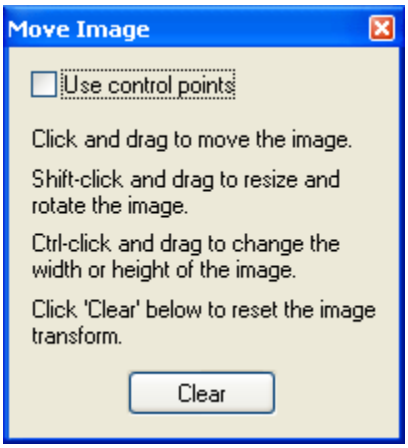
Define the units, datum, and projection for the image coordinate system using the image import wizard. The last page of the wizard asks you to define georeferencing information. The page should default to a “matrix reference” with all the values set to zero. Click “Finish” to complete the import without georeferencing information.

Once the image is read, it will show up in the layer view, but not in the map view. Right-click on the image and choose “Move Image” from the menu. This will bring up the Move Image tool in its initial state, shown below.



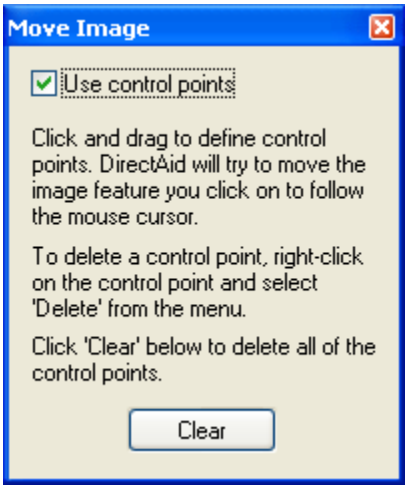
As the tool indicates, you can click and drag in the map view to define a rectangle for the image. The point you click on will be the top-left corner of the image, and the point you drag to will be the bottom-right, as shown below. The transform is updated in real time so you can see the effect of your movement.

With the initial placement complete, the Move Image tool changes, as shown below.



The image can be moved, rotated, and resized using simple mouse actions. Rotation occurs about the fixed point, which can be changed by clicking on the image. The fixed point is indicated in the map view by a circle and crosshairs. Any single action in the Move Image tool can be undone/redone as usual.

Once the image is roughly in place, you may want to refine the transform using control points. To enter control points, click “Use control points” in the Move Image tool. The tool will change as shown below.



To define control points, click on an image feature and drag it to the place where you want it to be in the map view. Control points are indicated by a circle and crosshairs, as shown below.

If you have defined three or fewer control points, then DirectAid will be able to match all control points exactly using a linear transform. If you define four or more control points, DirectAid may not be able to put the image feature exactly where you indicate, In this case the circle and crosshairs may not line up exactly, as shown below.

The circle indicates the position of the map feature, and the crosshairs indicates where you are trying to put it. The distance between the two is the transform error. If you have a significant distance between the two, there are a few possible causes:

- You may have picked the wrong image feature or position. In this case, you will need to delete and redefine the control point as described below.
- You may have picked the wrong coordinate system for the image. The linear transform used by DirectAid cannot, for example, squeeze an image at the top and stretch it at the bottom (an effect commonly known as keystoning). If it appears that you need this kind of transform, you may need to switch the image coordinate system from "UTM" to "latitude/longitude", for example.

As mentioned above, it is possible to delete control points which have been entered. If you have just finished entering an incorrect control point, it is probably fastest to undo the operation. Otherwise, place the mouse over a control point. It should appear highlighted, as shown below. Right-click on the control point and choose "Delete" from the menu. The transform should be updated immediately.

If you have defined control points, and then subsequently uncheck the "Use control points" box and re-transform the image, you will lose your defined control points. This is because control points generally tie an image feature to a position in the real world. When the image is re-transformed by hand, there is no reliable way to adjust the control points.

Image properties

To change image properties, right-click on the image in the layer view and choose *Properties* from the context menu. The *Properties* dialog contains the usual three tabs for coordinate system information, as well as a *Georeferencing* tab and a *Clipping* tab.

Image georeferencing

The *Georeferencing* tab in the image properties dialog specifies the conversion from image coordinates (measured in image pixels) to the "source" coordinate system (specified by the three standard coordinate system tabs). When an image is imported, this information is usually obtained either from GeoTIFF tags, a TFW file, or an OziExplorer MAP file. An image may be georeferenced using control points in either geodetic coordinates (lat/long) or grid coordinates (e.g. UTM or State Plane coordinates), or using a specific linear transform. In the latter two cases, the control points are converted to a linear transform. To select a method, choose from the drop-down list in the bottom-right corner of the page.

Image clipping

At this time, the only part of an image layer which can be edited outside its *Properties* dialog box is the clipping boundary. To edit the image clipping boundary, change to map object selection mode and double-click the image. The clipping boundary will be drawn with a transparent overlay and a thick box at each node of the boundary. To edit the boundary, you can either click and drag the nodes, or create a new node by clicking on an edge. To remove a node, right-click on the node and select *Delete* from the context menu.

An image clipping boundary is defined not only by the nodes, but also by the curve that the edges follow between the nodes. In particular, it is common for the edges to follow one of two curves. The curve may follow a line which is straight in grid coordinates, or it may follow a line which is straight in geodetic latitude/longitude. The curve may also follow a line which is straight in pixel coordinates, but this is usually just a special case of one of the previous two options. To change the clipping mode, go to the image properties by right-clicking on the image either in the layer view or in the map view, and choosing *Properties* from the context menu. Remember that the context menu can be accessed from the map view only if you are in map object selection mode. Choose the *Clipping* tab. You should see something like Figure 128.

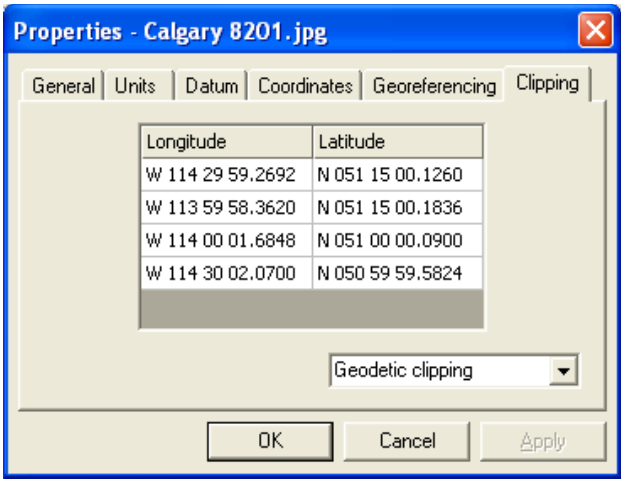


Figure 128 - Image clipping

The drop-down box in the bottom-right indicates which clipping mode is currently active. To change the clipping mode, select a new mode from the drop-down. Note that the vertices of the clipping polygon are specific to a particular clipping mode, so the vertex list for some modes may be blank. At this time it is not possible to add or remove vertices from this interface, although it is possible to edit the vertex positions. When you have finished, click *OK* to close the dialog box.

Image Transparency

To change an image's transparency level, open its Properties and change to the *General* tab. Adjust the slider labeled "Opacity". This can be especially useful when you're georeferencing an image against a background image.

Printing and Print Preview

To print the current workspace to a printer using the same centre and scale as in the map view, choose *Print* from the *File* menu, or use the toolbar button (🖶).. This will bring up the standard Windows print dialog, where you can choose a printer, set printer properties, and set the number of copies. To complete the operation, click *OK*.

As usual, page and printer options can be changed using the *Page Setup* command, located in the *File* menu. To preview the print, choose *Print Preview* from the *File* menu. By clicking with the left mouse button in the preview area, the user can cycle through three standard zoom settings.

Printing to a File

Printing a whole page

There are two methods of printing to a file in DirectAid. The first is analogous to printing to the printer. This method will result in a file which has the same centre as the current map view, but with a user-defined scale, page size and resolution. To print using this method, choose *Print To File* from the *File* menu. You will be prompted for a file name and type. DirectAid 2.0 will add the appropriate extension to the file name if you do not enter one yourself. When you have done, click *Save* to continue. This will bring up the first page of the print -to-file wizard, as shown in Figure 129.

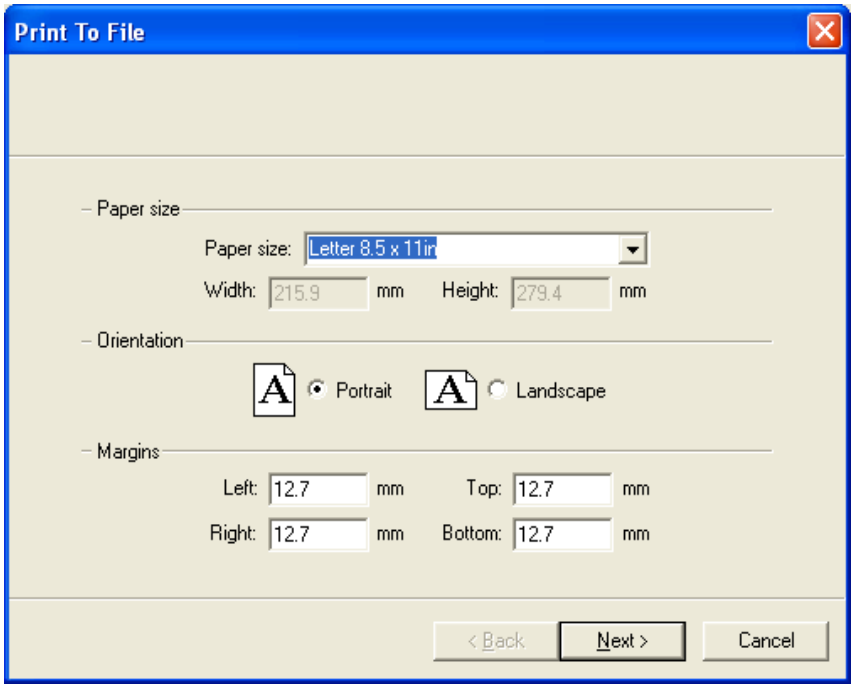


Figure 129 - Printing a whole page (step 1)

On the first page of the wizard, you are asked to choose the page dimensions. You may choose from three standard paper sizes, or you may specify a custom paper size. If you choose the former, you may also choose to print in portrait or landscape orientation. Finally, you may adjust the page margins. The output file will include only the page area within these margins. When you are satisfied with your choices, press *Next*. This will bring up the second page of the wizard, as shown in Figure 130.

On the second page of the wizard, you are asked to choose the map scale and image dimensions. If your camera coordinate system is geodetic latitude/longitude, this scale will be measured in degrees per page metre. Otherwise, the scale will be measured in units on the ground per unit on the page, as in standard maps.

There are three choices for determining image dimensions. First, you may specify a world scale, measured in metres on the ground per image pixel. This is typical for images which will be read back into mapping software, where the resolution you have in mind is measured on the ground. Second, you may specify a page scale, either in dots per page inch, or page millimeters per pixel. This is useful if you intend to send the image to a printer, and have a target resolution in mind. Finally, you may choose to specify the size of the image directly. You might use this option if you are planning to send the image by e-mail and have a particular image size in mind. In either of the three cases, the inactive options will be updated as you edit the active option, so that you can perform a "reality check" on the numbers.

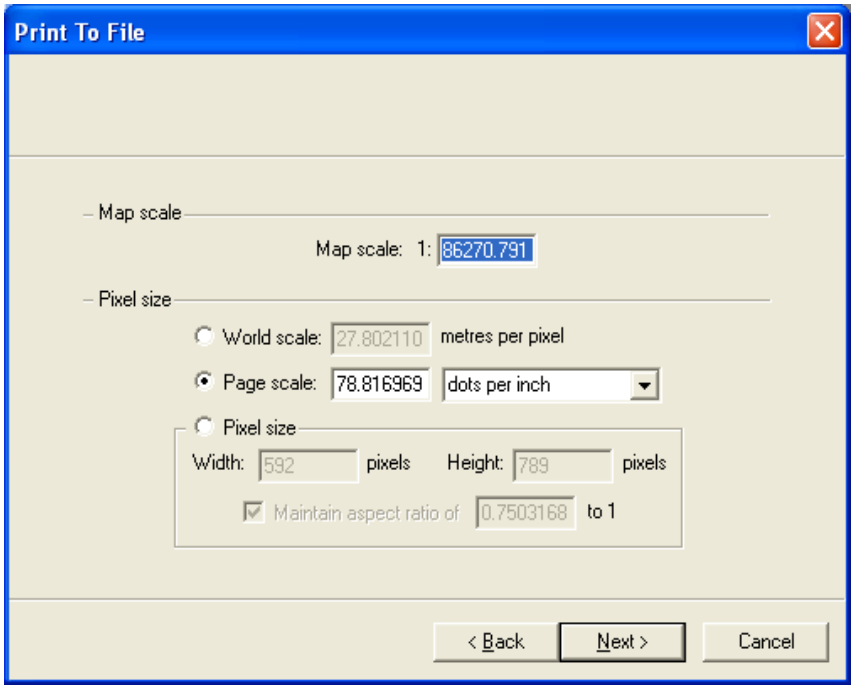


Figure 130 - Printing a whole page (step 2)

In some cases, this is all there is to the wizard. In other cases (for example when writing a JPEG file), there may be further pages where you may specify parameters specific to the image format. When you are done, click *Finish* to generate the image.

In addition to the image file itself, DirectAid will generate an auxiliary OziExplorer MAP file and TFW file. If you are writing a TIFF image, DirectAid will also write GeoTIFF tags to the output file. The output image will be referenced in the camera coordinate system.

Printing a region

The second printing method is used if you want to print a particular part of the map view to an image file. First, select the page-area selection tool, either by choosing *Page area* from the *Selection* menu, or using the toolbar button ( ). Now select an area of the map

view by clicking and dragging across the map. The selected area will be bounded by an animated box. To clear this selection when you are done, simply click anywhere in the map without dragging. The page-area selection does not clear when you switch back to a different tool (for example, the pan or zoom tools).

Once you have selected your desired image area, once again choose *Print to File* from the *File* menu. You will be prompted for a file name and type. DirectAid 2.0 will add the appropriate extension to the file name if you do not enter one yourself. When you have done, click *Save* to continue. This will bring up a different print-to-file wizard, as shown in Figure 131.

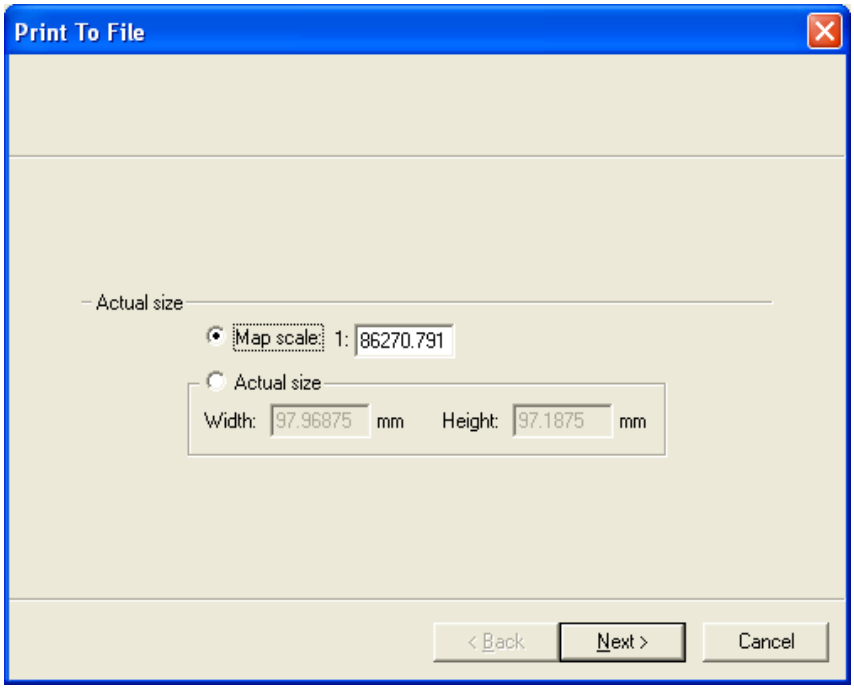


Figure 131 - Printing a region

On this page you are essentially being asked to specify the page dimensions. Whereas with the first method the area covered by the image could vary depending on page size and scale, in this case we have fixed the coverage area by selecting it within the map view. We can either specify the map scale, in which case the page size will be updated automatically, or we can specify the page size in millimetres, in which case the map scale is calculated. When you are done, click *Next* to continue.

The second page of this wizard is very similar to the second page of the wizard shown in Figure 130, except that map scale has been moved to the first page because of its effect on page size. Other than this small change, the second page of the wizard is still used to determine image dimensions. In some cases, again, there may be additional pages in the wizard, requesting parameters specific to the image format. When you are done, click *Finish* to generate the image. As before, auxiliary georeferencing information will also be written.

This method of printing to a file can be used to generate image mosaics using source images in any coordinate system. First, set the camera coordinate system to your desired output datum and projection. It may be necessary to re-square your camera axes () so that the output image is square to the desired coordinate system. Second, import whatever images you would like to include in the mosaic. Finally, select the area to be output in the mosaic and use *Print to File* to write the image. When generating image mosaics, the "map scale" and "actual size" on the first wizard page will have no effect on the output. These parameters affect only the size of symbols and lines, whose thickness is referenced in page coordinates.

Version Control

With DirectAid 2.0, we have taken the concept of "undo/redo" a bit further than you may be used to. The program logs every change made to a workspace. There are two main methods of working with this log: *undo/redo* and *reverting a layer*. Collectively, we refer to these methods as *version control*.

Undo/redo

The simplest form of version control is the undo/redo functionality you are probably already familiar with. To undo actions in the reverse order in which they were done, either choose *Undo* from the *Edit* menu, or press *Ctrl-Z*. To redo an action which has just been undone, choose *Redo* from the *Edit* menu, or press *Ctrl-Y*. Although most actions can be undone, there are a few (for example, deleting an entire project) which cannot be undone because a large amount of data would need to be stored in order to ensure the action could be undone. If you perform such an action, DirectAid will display a warning dialog asking if you would like to save the data to disk before modifying it.

Reverting a layer

In addition to undoing changes in order, in some cases it is also possible to undo changes just to a part of the workspace (a single layer and all the layers it contains). There are two ways to do this. The first method, a *partial undo*, can be used to roll back a series of changes made to a particular layer (or its contents). The second method, a *revert*, is more like a "time machine". It allows you to bring the layer (and its contents) back to some previous state. These states are usually identified by *bookmarks*.

Partial undo

The simplest way to revert a part of the project is to undo only changes done to that part of the project. For example, suppose you have made many changes to a project, but wish to undo only a few changes made to a particular line. First, right-click on the line in the layer view and choose *Undo/Revert* from the context menu. This will bring up the *Undo/Revert Tool*, shown in Figure 132. By default, the undo/revert tool shows a list of changes made to the layer (or any layer it contains). The most recent changes are shown at the top of the list.

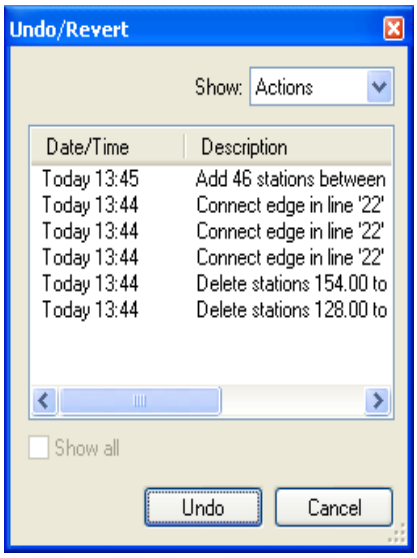


Figure 132 - The undo tool

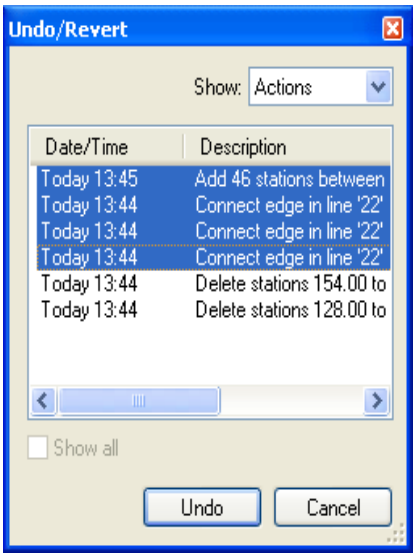


Figure 133 - The undo tool (with selection)

To undo changes, click on a single change. All changes above the selected change will also be selected, since it is impossible to undo one change without undoing everything that came after. With a selection made, click *Undo* to make the change.

Bookmarks

Before describing the process of reversion, it is important to describe how we will know "where" to go back to. Although each state resulting from a change to the workspace is marked with the date and time, this is clearly not the best way to look back. After all, do you really know if the state you want to go to was the created at 9:30 this morning, or at 9:35? For this reason, we have created *bookmarks*.

A bookmark is a description you can assign to a particular workspace state. For example, after you have created a new project, you might create a bookmark called "Created new project". After you have trimmed the project, you might create another one called "Trimmed the project". When you have moved a few lines to existing trail, you might create a third bookmark called "Move to existing trail". If you later decide that you want to put a particular receiver line back the way it was just after you trimmed the project, you could revert that line to the bookmark called "Trimmed the project".

To create or modify a bookmark, either choose *Bookmark* from the edit menu, or use the shortcut key *Ctrl-B*. If the *Edit Bookmark* dialog does not appear, check that the main window is active, and not a tool window. Enter a description for the bookmark, then click *OK* or hit *Enter*. We've made this as easy as possible in order to encourage you to use bookmarks often. It is an excellent idea to get into the habit of creating a new bookmark after you have made a series of related changes to the project.

Reverting

Reverting is similar to the partial undo above, except that instead of specifying a group of changes to be undone, you will select a bookmark to "revert" to. To revert a layer, right-click on the layer in the layer view and choose *Undo/Revert* from the context menu. In the drop-list labeled *Show*, select *Bookmarks*. The list below changes to show all the bookmarks to which the selected layer can be reverted, as shown in Figure 134.

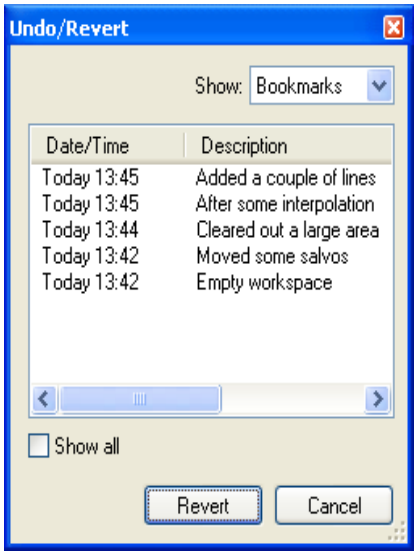


Figure 134 - The revert tool

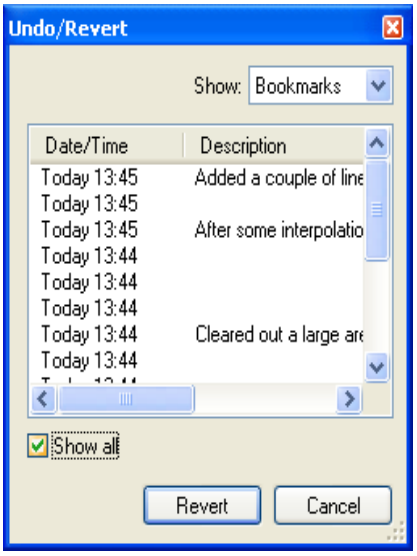


Figure 135 - The revert tool (expanded)

By default, the revert tool shows only those states which have been bookmarked. To show every state, check the box marked *Show all*. The state list will be expanded as shown in Figure 135. In general, it is not advisable to revert to an unmarked state, since it is difficult to tell exactly what changes will be made. The complete list is provided in case you should forget to bookmark a state, but are lucky enough that the time stamp tells you what you need to know.

With a selection made, click *Revert* to complete the action. The selected layer (and its contents) will be rolled back to the bookmark you chose. The revert itself becomes the most recent action. If you need to undo the revert action, for example if you reverted to the wrong place, just press *Ctrl-Z*.

Appendix A DirectAid Extensions to Python

This section will be of interest mainly to users who are familiar with the Python programming language, and would like to build their own reports. In addition to the Python standard library, the reports supplied with DirectAid make use of the following free libraries:

- The ReportLab Toolkit (http://www.reportlab.org/rl_toolkit.html) is used as a framework for generating PDF documents.
- The Python Imaging Library (http://www.pythonware.com/products/pil/) is used for image manipulation.
- NumPy (http://numpy.scipy.org/) is used for numerical analysis.
- Matplotlib (http://matplotlib.sourceforge.net/) is used for histogram plots.

Any user who is looking to design their own reports should consider familiarizing themselves with these packages.

To generate reports, DirectAid launches an embedded instance of the Python interpreter. On initialization, several DirectAid-related modules are automatically added to the Python namespace. These modules give Python a way of interacting with DirectAid workspaces.

In addition to this "low-level" interface, we have built a framework for generating PDF reports, based on the free libraries listed above. We have used this framework to build several reports which are included with DirectAid.

Following is a reference describing some of the more important DirectAid extensions for Python.

The low-level interface

The modules, classes and functions associated with the low-level Python interface are outlined in Figure 136 below.

The `directaid` module contains a single class: `Property`. This class is very important, since it is the main interface between Python and the DirectAid workspace.

All DirectAid layers are exposed as a `Property` item within Python. DirectAid provides interfaces to access two layers directly: The active project and midpoint layers. Children of these layers can be accessed using child `Property` items, which are accessed using a dictionary-like interface. Child `Property` items include some built-in properties, and also the user-defined properties for that layer. For example, the following code iterates through all of the source lines, displaying their line number:

```
from directaid import project
root = project.root()
```

```
sources = root['_SOURCES']
for line in sources:
     print 'Source line %s' % line['_LABEL']
```

The remainder of the modules comprising the low-level Python interface to DirectAid are described below.

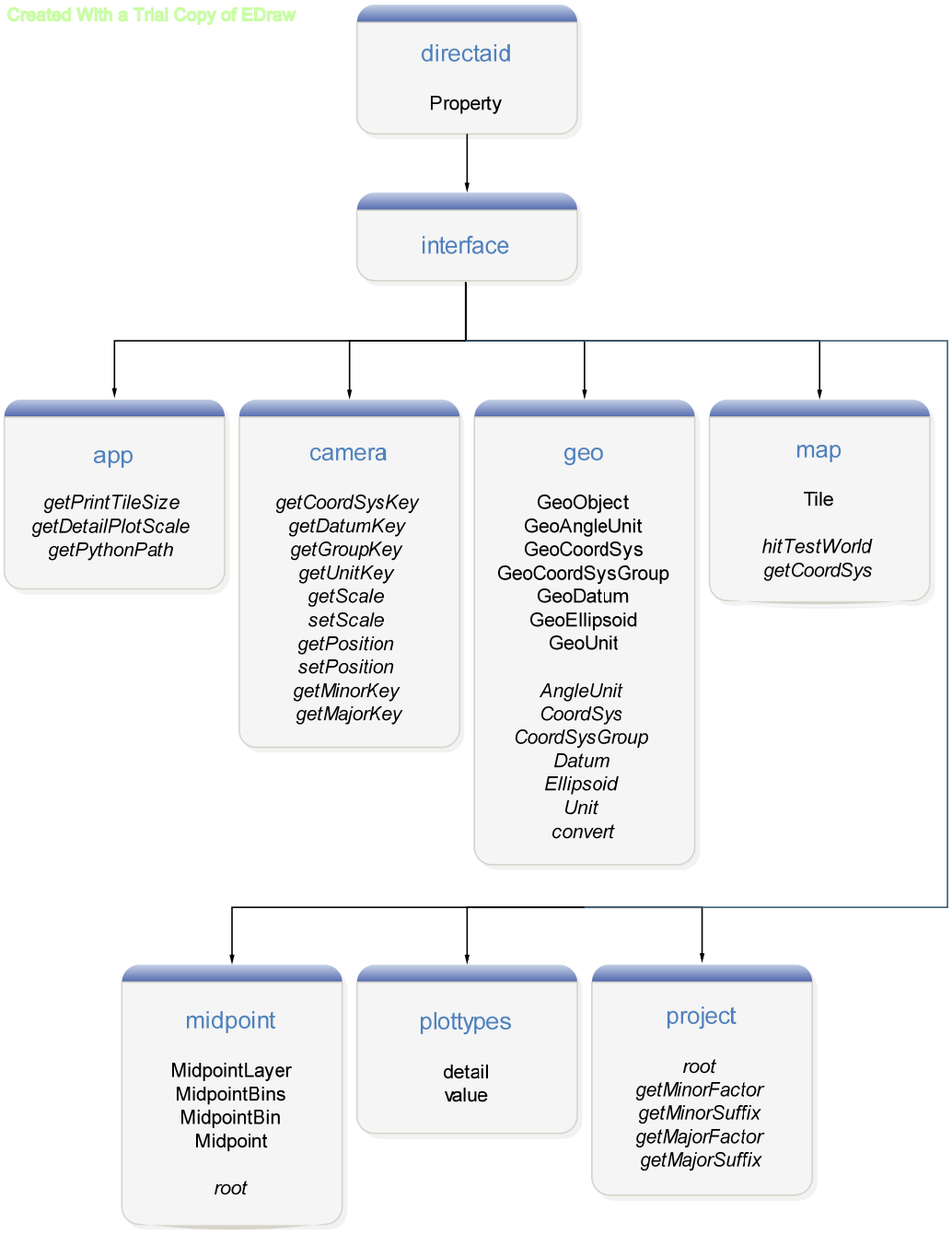


Figure 136 - The low-level Python interface

directaid.interface.app

getPrintTileSize ()

Returns the size, in pixels, of image tiles used for printing.

getDetailPlotScale ()

Returns scale at which detail plots become visible.

getPythonPath ()

Returns the path for DirectAid's Python extensions.

directaid.interface.camera

getCoordSysKey()

Returns the camera's coordinate system key.

getDatumKey()

Returns the camera's datum key.

getGroupKey()

Returns the camera's coordinate system group key.

getUnitKey()

Returns the camera's linear unit key.

getScale()

Returns the camera's scale.

setScale(scale)

Parameters

`scale` The camera's new scale.

Sets the camera scale.

getPosition()

Returns the camera's angular unit key.

setPosition(x, y, z)

Parameters

`x, y, z` The camera's new position.

Sets the camera position.

getMinorKey()

Returns the camera's minor linear unit key.

getMajorKey()

Returns the camera's major linear unit key.

directaid.interface.geo

class GeoObject

`getName()`

Returns the name of the object.

`getKey()`

Returns the object's key string.

class GeoAngleUnit(GeoObject)

class GeoCoordSys(GeoObject)

class GeoCoordSysGroup(GeoObject)

class GeoDatum(GeoObject)

class GeoEllipsoid(GeoObject)

class GeoUnit(GeoObject)

`suffix()`

Returns the unit's suffix.

`factor()`

Returns the conversion factor to metres.

AngleUnit(key)

Parameters

`key` The key string for the angular unit.

Returns the angular unit object.

CoordSys(key)

Parameters

`key` The key string for the coordinate system.

Returns the linear coordinate system.

CoordSysGroup(key)

Parameters

`key` The key string for the coordinate system group.

Returns the linear coordinate system group.

Datum(key)

Parameters

`key` The key string for the datum.

Returns the linear datum.

Ellipsoid(key)

Parameters

`key` The key string for the ellipsoid.

Returns the linear ellipsoid.

Unit(key)

Parameters

`key` The key string for the linear unit.

Returns the linear unit object.

convert(csIn, posIn, csOut)

Parameters

csIn	The input coordinate system.
posIn	The input position.
csOut	The output coordinate system.

Returns the converted position.

directaid.interface.map

class Tile

```
__init__(self, cxStrip, cyStrip, cxPage, cyPage,
    mmWidth, mmHeight)
```

Parameters

cxStrip	Width of the tile in pixels.
cyStrip	Height of the tile in pixels.
cxPage	Width of the map in pixels.
cyPage	Height of the map in pixels.
mmWidth	Width of the tile in millimeters on the printed page.
mmHeight	Height of the tile in millimeters on the printed page.

```
getImage(self, x, y, wTile, hTile)
```

Parameters

x	Position of left edge of tile, in millimeters from the left edge of the printed page.
y	Position of top edge of tile, in millimeters from the bottom edge of the printed page.
wTile	Width of the tile in millimeters on the printed page.

`hTile` Height of the tile in millimeters on the printed page.

Returns an image property object containing the rendered map tile.

`hitTestWorld(pos)`

Parameters

`pos` Page position to test, in millimeters, measured from bottom-left corner of the page.

Returns a tuple containing the 3-dimensional world position which lies under the test point and on the camera datum.

`getCoordSys()`

Returns the camera's coordinate system key.

`directaid.interface.midpoint`

`root()`

Returns the active midpoint layer.

`class MidpointLayer`

`histogram(self, args)`

Parameters

`args` A list of tuples, each consisting of plot type, lower limit, upper limit, and bin count. For example: `[('fold', 1, 20, 20)]`.

Returns a numeric array containing the histogram data.

`limits(self, key)`

Parameters

`key` The plot type, e.g. `'fold'`.

Returns a tuple containing the minimum and maximum value for the specified plot type.

directaid.interface.plottypes

value

`null`	No value plot
`fold`	Fold
`offsetGap`	Largest offset gap
`offsetDispersion`	Offset dispersion
`offsetSqGap`	Offset-squared gap
`offsetSqDispersion`	Offset-squared dispersion
`azimuthGap`	Azimuth gap
`azmiuthDispersion`	Azimuth dispersion

detail

`null`	No detail plot
`midpoint`	Midpoint distribution
`offset`	Offset distribution
`offsetSq`	Offset-squared distribution
`azimuth`	Azimuth distribution
`offsetAzimuth`	Offset-azimuth distribution

directaid.interface.project

root()

Returns the active project.

getMinorFactor()

Returns the factor to convert from project units to the camera's minor unit.

getMinorSuffix()

Returns the suffix for camera's minor unit.

getMajorFactor()

Returns the factor to convert from project units to the camera's major unit.

getMajorSuffix()

Returns the suffix for camera's major unit.

The PDF report framework

In principle, the low-level Python interface described above can be used in combination with any Python library to produce output of almost unlimited variety. Because users will very often want to produce reports in PDF format, we have assembled a framework, built around the ReportLab Toolkit and other freely available Python libraries, for doing just that. The modules, classes and functions comprising the PDF report framework are outlined in Figure 137 below.

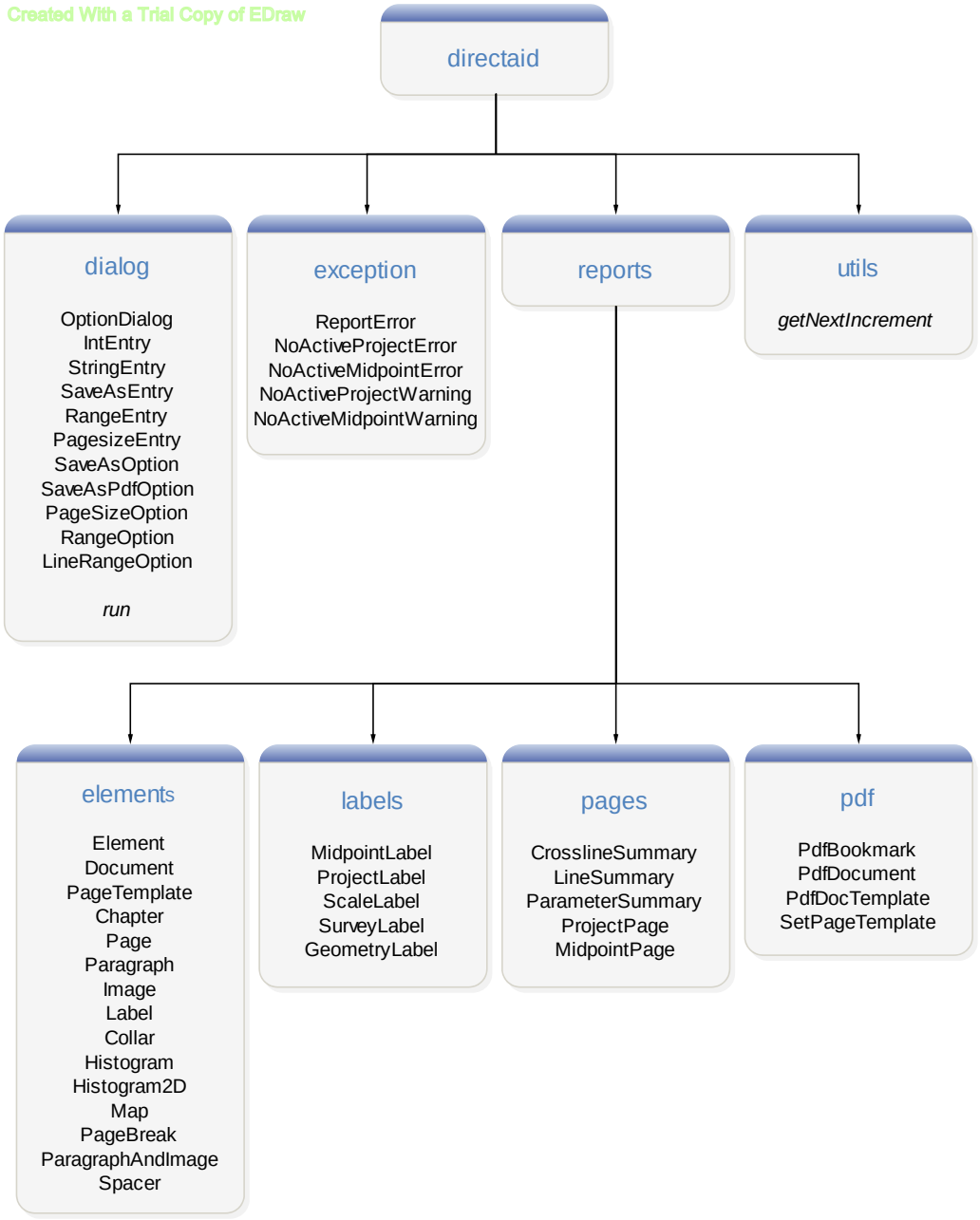


Figure 137 - The PDF report framework

directaid.dialog

class OptionDialog

`__init__(self, options, title='Report Options')`

Parameters

`options`	Report options.
`title`	Dialog title.

`onCancel(self)`

Called when “Cancel” button is pressed.

`onOK(self)`

Called when “OK” button is pressed.

`destroy(self)`

Destroys the dialog box.

`run(self)`

Shows the dialog box.

`setPosition(self, width, height)`

Parameters

`width`	Width of the dialog box in pixels.
`height`	Height of the dialog box in pixels.

Centers the dialog box on the screen.

`getOption(self, key)`

Parameters

`key`	The option key.

Returns the option value.

`getOptions(self)`

Returns a dictionary of all options.

`addOptions(self, options)`

Parameters

`options`	A dictionary of options to add.

Adds the options to the dialog's dictionary.

`class IntEntry(Tile.Frame)`

`__init__(self, master, row, label, val)`

Parameters

`master`	The parent widget.
`row`	Row in parent widget.
`label`	Text label.
`val`	Initial value.

`getValue(self)`

Returns the wrapped value.

`validate(self)`

Validates the wrapped value.

`class StringEntry(Tile.Frame)`

`__init__(self, master, row, label, val)`

Parameters

`master`	The parent widget.
`row`	Row in parent widget.
`label`	Text label.
`val`	Initial value.

`getValue(self)`

Returns the wrapped value.

`validate(self)`

Validates the wrapped value.

`class SaveAsEntry(Tile.Frame)`

`__init__(self, master, row, label, filename, extension)`

Parameters

`master`	The parent widget.
`row`	Row in parent widget.
`label`	Text label.
`filename`	Default filename.
`extension`	Default extension.

`onSaveAs(self)`

Called when “Save As” button is pressed.

`getValue(self)`

Returns the wrapped filename.

`validate(self)`

Validates the wrapped filename.

`class RangeEntry(Tile.Frame)`

`__init__(self, master, row, label, vals)`

Parameters

`master`	The parent widget.
`row`	Row in parent widget.
`label`	Text label.

`vals` A tuple consisting of minimum value, maximum value, value step, and unit suffix.

`getValue(self)`

Returns a tuple consisting of minimum value, maximum value, and value step..

`validate(self)`

Validates the wrapped range.

`class PagesizeEntry(Tile.Frame)`

`__init__(self, master, row, label)`

Parameters

`master` The parent widget.

`row` Row in parent widget.

`label` Text label.

`getValue(self)`

Returns the wrapped pagesize.

`validate(self)`

Validates the wrapped pagesize.

`class SaveAsOption`

`__init__(self, filename, extension)`

Parameters

`filename` Default filename.

`extension` Default extension.

`addDlgEntry(self, dlg, label)`

Parameters

`dlg` Parent dialog box.

`label` Text label.

Adds the a “Save As” entry to the dialog.

`class SaveAsPdfOption(SaveAsOption)`

`__init__(self, filename)`

Parameters

`filename` Default filename.

`class PageSizeOption`

`__init__(self, size)`

Parameters

`size` Default pagesize.

`addDlgEntry(self, dlg, label)`

Parameters

`dlg` Parent dialog box.

`label` Text label.

Adds the a pagesize entry to the dialog.

`class RangeOption`

`__init__(self, minimum, maximum, step, unit=None)`

Parameters

`minimum` Minimum value.

`maximum` Maximum value.

`step` Value step.

`unit` Unit suffix.

`addDlgEntry(self, dlg, label)`

Parameters

`dlg` Parent dialog box.

`label` Text label.

Adds the a range entry to the dialog.

`class LineRangeOption(RangeOption)`

`__init__(self, minimum, maximum, step)`

Parameters

`minimum` Minimum line number.

`maximum` Maximum line number.

`step` Line number increment.

`class ProgressDialog`

`__init__(self, title='Report')`

Parameters

`title` Dialog title.

`destroy(self)`

Destroys the dialog box.

`setPosition(self)`

Centers the dialog box on the screen.

`run()`

Shows the dialog box.

directaid.exception

`class ReportError(Exception)`

`__init__(self, message)`

Parameters

`message` Error message.

class NoActiveProjectError(ReportError)

`__int__(self)`

class NoActiveMidpointError(ReportError)

`__int__(self)`

NoActiveProjectWarning()

NoActiveMidpointWarning()

directaid.reports.elements

class Element

`__init(self)`

`options(options)`

Parameters

`options`	Report options.

Adds options for this element to the `options` list.

`getFlowables(self)`

Returns a list of ReportLab flowable objects.

class Document(Element)

`__init__(self, filename, pagesize)`

Parameters

`filename`	Name of the file the document will be saved to.
`pagesize`	A tuple containing width, height of the page.

`add(self, chapter)`

Parameters

`chapter`	A chapter to be added to the document.

Adds a chapter to the document.

`getPrintableSize(self, template)`

Parameters

`template` Page template.

Returns a tuple containing width, height of the printable area of the page for this page template (not including margins).

`getFlowables(self)`

Returns a list of ReportLab flowable objects.

class PageTemplate

```
__init__(self, pagesize, leftMargin=0,
   rightMargin=0, topMargin=0, bottomMargin=0)
```

Parameters

`pagesize`	A tuple containing width, height of the page.
`leftMargin`	Left margin, in millimetres.
`rightMargin`	Right margin, in millimetres.
`topMargin`	Top margin, in millimetres.
`bottomMargin`	Bottom margin, in millimetres.

`width(self)`

Returns the width of the prindtable area (not including margins).

`height(self)`

Returns the height of the prindtable area (not including margins).

class Chapter(Element)

```
__init__(self, doc, template=None,
   parentTemplate=None, title=None)
```

Parameters

`doc`	Parent document.

`template`	Page template for this chapter.
`parentTemplate`	Page template for the parent chapter.
`title`	Chapter title.

`add(self, element)`

Parameters

`element`	Element to be added to the chapter.

Adds an element to the chapter.

`getFlowables(self)`

Returns a list of ReportLab flowable objects.

`class Page(Element)`

`__init__(self, pagesize, title=None)`

Parameters

`pagesize`	Size of the page.
`title`	Page title.

`add(self, element, pos, align='bl')`

Parameters

`element`	Element to be added to the page.
`pos`	A tuple containing the x, y position of a corner of the element.
`align`	Specifies which corner lies at `pos`. Available options are `'tr'`, `'tl'`, `'br'`, and `'bl'`, which correspond to the top-right, top-left, bottom-right, and bottom-left corners, respectively.

Adds and element to the page.

`getFlowables(self)`

Returns a list of ReportLab flowable objects.

class Paragraph(Element)

```
__init__(self, text, fontsize,
   fontname='Helvetica', alignment='centre')
```

Parameters

`text`	The text of the paragraph.
`fontsize`	Size of the font (point size).
`fontname`	Name of the font.
`alignment`	Text alignment. Available options are `'left'`, `'right'`, and `'centre'`.

`getFlowables(self)`

Returns a list of ReportLab flowable objects.

class Image(Element)

```
__init__(self, data, width, height=None,
   iwidth=None, iheight=None, mode='RGB')
```

Parameters

`data`	Either an image filename, or raw pixel data for the image. If this parameter is a filename, `iwidth`, `iheight`, and `mode` are ignored.
`width`	Image width in millimeters.
`height`	Image height in millimeters.
`iwidth`	Image width in pixels.
`iheight`	Image height in pixels.
`mode`	Image mode, as documented in `PIL.Image.frombuffer`.

`getFlowables(self)`

Returns a list of ReportLab flowable objects.

class Label(Element)

`__init__(self, data, width, height, border=True, grid=False, style=None)`

Parameters

`data`	Write something here.
`width`	Label width in millimeters.
`height`	Label height in millimeters.
`border`	Draw a border around the label.
`grid`	Draw a grid between label entries.
`style`	A ReportLab `TableStyle` object specifying optional style parameters for the label. This is documented in `reportlab.platypus.TableStyle`.

`getFlowables(self)`

Returns a list of ReportLab flowable objects.

class Collar(Element)

`__init__(self, width, height, size=0.5*inch, dpi=300)`

Parameters

`width`	Width of the box containing the collar in millimeters.
`height`	Height of the box containing the collar in millimeters.
`size`	Thickness of the collar in millimeters.
`dpi`	Resolution of the collar.

`clientArea(self)`

Returns a tuple containing the width, height of the inner box of the collar..

`getFlowables(self)`

Returns a list of ReportLab flowable objects.

`class Histogram(HistogramBase)`

```
__init__(self, width, height, mp, xaxis, heading,
  align='edge', factor=1, suffix='', dpi=300)
```

Parameters

`width`	Width in millimeters.
`height`	Height in millimeters.
`mp`	A midpoint layer.
`xaxis`	Label for the horizontal axis.
`heading`	Histogram label.
`align`	Value alignment. To align at the left edge of the bar, use `'edge'`. To align in the centre of the bar, use `'centre'`.
`factor`	Factor to multiply histogram units by for display.
`suffix`	Unit suffix.
`dpi`	Resolution of the histogram.

`class Histogram2D(HistogramBase)`

```
__init__(self, width, height, mp, xaxis, yaxis,
  heading, align='edge', xfactor=1, yfactor=1,
  xsuffix='', ysuffix='', dpi=300)
```

Parameters

`width`	Width in millimeters.
`height`	Height in millimeters.

`mp`	A midpoint layer.
`xaxis`	Label for the horizontal axis.
`yaxis`	Label for the vertical axis.
`heading`	Histogram label.
`align`	Value alignment. To align at the left edge of the bar, use `'edge'`. To align in the centre of the bar, use `'centre'`.
`xfactor`	Factor to multiply horizontal units by for display.
`yfactor`	Factor to multiply vertical units by for display.
`xsuffix`	Horizontal unit suffix.
`ysuffix`	Vertical unit suffix.
`dpi`	Resolution of the histogram.

`class Map(Element)`

`__init__(self, size, dpi=300)`

Parameters

`size`	A tuple containing width, height of the map, in millimeters.
`dpi`	Resolution of the map, in pixels per inch.

`getFlowables(self)`

Returns a list of ReportLab flowable objects.

`class PageBreak(Element)`

`__init__(self)`

`getFlowables(self)`

Returns a list of ReportLab flowable objects.

class ParagraphAndImage(Element)

`__init__(self, paragraph, img, side='left')`

Parameters

`paragraph`	A Paragraph element.
`img`	An Image element.
`side`	The side the image goes on. Available options are `'left'` or `'right'`.

`getFlowables(self)`

Returns a list of ReportLab flowable objects.

class Spacer(Element)

`__init__(self, width, height)`

Parameters

`width`	Width of the spacer in millimeters.
`height`	Height of the spacer in millimeters.

`getFlowables(self)`

Returns a list of ReportLab flowable objects.

directaid.reports.labels

class MidpointLabel(Label)

`__init__(self, width, fontsize)`

Parameters

`width`	Width of the label in millimeters.
`fontsize`	Size of the font (point size).

Returns write something here.

class ProjectLabel(Label)

`__init__(self, width, fontsize)`

Parameters

`width` Width of the label in millimeters.

`fontsize` Size of the font (point size).

Returns write something here.

`class ScaleLabel(Label)`

`__init__(self, width, units, fontsize)`

Parameters

`width` Width of the label in millimeters.

`units` A list of linear unit keys to be displayed in the scale bar. For example, `['KILOMETERS', 'U.S. SURVEY MILES']`.

`fontsize` Size of the font (point size).

Returns write something here.

`class SurveyLabel(Label)`

`__init__(self, width, text, srv, fontsize)`

Parameters

`width` Width of the label in millimeters.

`text` Survey layer type, e.g. `'Source'` or `'Receiver'`.

`srv` The survey layer the label describes.

`fontsize` Size of the font (point size).

Returns write something here.

`class GeometryLabel(Label)`

`__init__(self, width, fontsize)`

Parameters

`width`	Width of the label in millimeters.
`fontsize`	Size of the font (point size).

Returns write something here.

directaid.reports.pages

class CrosslineSummary(Chapter)

```
__init__(self, doc, options, width, template=None,
   parentTemplate=None)
```

Parameters

`doc`	Parent document.
`options`	Report options.
`width`	Label width in millimeters.
`template`	Page template for this chapter.
`parentTemplate`	Page template for parent chapter.

Returns write something here.

```
options(options)
```

Parameters

`options`	Report options.

Adds options for this element to the `options` list.

class LineSummary(Chapter)

```
__init__(self, doc, options, width, survey,
   template=None, parentTemplate=None, title='Line
   Summary')
```

Parameters

`doc`	Parent document.
`options`	Report options.

`width`	Label width in millimeters.
`survey`	Survey layer for line summary.
`template`	Page template for this chapter.
`parentTemplate`	Page template for parent chapter.
`title`	Chapter title.

`options(options)`

Parameters

`options`	Report options.

Adds options for this element to the `options` list.

`class ParameterSummary(Chapter)`

```
__init__ (self, doc, options, width,
   template=None, parentTemplate=None)
```

Parameters

`doc`	Parent document.
`options`	Report options.
`width`	Label width in millimeters.
`template`	Page template for this chapter.
`parentTemplate`	Page template for parent chapter.

Returns write something here.

`options(options)`

Parameters

`options`	Report options.

Adds options for this element to the `options` list.

class ProjectPage(Page)

```
__init__ (self, options, pagesize, scale=None,
   title=None)
```

Parameters

`options`	Report options.
`pagesize`	A tuple containing width, height of the page.
`scale`	Map scale (map units per page unit), or `None` to keep the current scale.
`title`	Page title.

```
options(options)
```

Parameters

`options`	Report options.

Adds options for this element to the `options` list.

class MidpointPage(ProjectPage)

```
__init__ (self, options, pagesize, mp, valuePlot,
   detailPlot, binGrid=False, scale=None,
   title=None)
```

Parameters

`options`	Report options.
`pagesize`	A tuple containing width, height of the page.
`mp`	Active midpoint layer.
`valuePlot`	See `plottypes` on page A-8.
`detailPlot`	See `plottypes` on page A-8.
`binGrid`	Draw bin grid.
`scale`	Map scale (map units per page unit), or `None` to keep the current scale.

`title` Page title.

directaid.reports.pdf

class PdfBookmark(ActionFlowable)

`__init__(self, title=None)`

Parameters

`title` Bookmark title.

`apply(self, doc)`

Parameters

`doc` Parent document.

Executes the action (see ReportLab documentation for ActionFlowable).

class PdfDocument(Document)

`__init__(self, options, title=None, author=None)`

Parameters

`options` Report options.

`title` Document title.

`author` Document author.

`options(options, filename='Report.pdf')`

Parameters

`options` Report options.

`filename` Default filename.

Adds options for this element to the `options` list.

`save(self)`

Writes the document.

`getValidPath(self, filename)`

Parameters

`filename` Base filename.

Appends a number to `filename`, if necessary, to ensure that the filename is unique.

`createTemplates(self)`

Returns a list of ReportLab PageTemplates.

`class PdfDocTemplate(BaseDocTemplate)`

`__init__(self, filename, pagesize, templates, title=None, author=None)`

Parameters

`filename` Document filename.

`pagesize` A tuple containing width, height of the page.

`templates` A list of ReportLab PageTemplates.

`title` Document title.

`author` Document author.

`build(self, flowables, onPage=_doNothing, canvasmaker=Canvas)`

Parameters

`flowables` List of ReportLab flowables.

`onPage` Function called whenever a new page is created..

`canvasmaker` Function that returns a `Canvas` object.

`handle_pageEnd(self)`

Called when a page is completed.

`beforeDocument(self)`

Called before flowables are processed.

`setNextPageTemplate(self, pt)`

Parameters

`pt` New page template.

`class SetPageTemplate(ActionFlowable)`

`__int__(self, pt)`

Parameters

`pt` New page template.

`apply(self, doc)`

Parameters

`doc` Parent document.

Executes the page template change.

`directaid.utils`

`getNextIncrement(inc)`

Parameters

`inc` The previous value in the sequence.

Returns the next value in the sequence 1, 2, 5, 10, 20, 50, …