Debrief

TMA Skills

This set of tutorials covers the skills necessary for single-sided track reconstruction - useful if one of your trial participants hasn't provided track data but for which you have sonar data.

www.debrief.info

Ian Mayo

Name: ____________________ Date: ________________

Please forward feedback on this tutorial to **support@debrief.info**

Single-Sided Reconstruction

In this tutorial, we will look at single-sided reconstruction. It's important to note that all analysis is done on tracks, and this involves both working with and analysing target tracks, including **grooming track data**, **grooming sensor data**, **generating**, and **merging tracks**. As such, you must know how to configure Debrief and how to load your data. So, before running through this tutorial, you should really have completed **Tutorial 3 - Controlling what you view**; if you haven't done this, please do so before attempting this tutorial.

Now it's time to start building and grooming the target track.

Grooming track data

This section will lead you through grooming some dodgy track data. First though, we need to load and understand the track data itself.

Load Data Files

To load the data files:

1. Open the **Navigator** view.
2. Expand the **SR2 folder** and locate the file **"dodgy_track.rep"**.
3. Double-click on it to load this track as a new Debrief plot.
4. Select **Over The Ground (OTG)** as the track mode, and the plot and associated views will promptly update.

Main track data

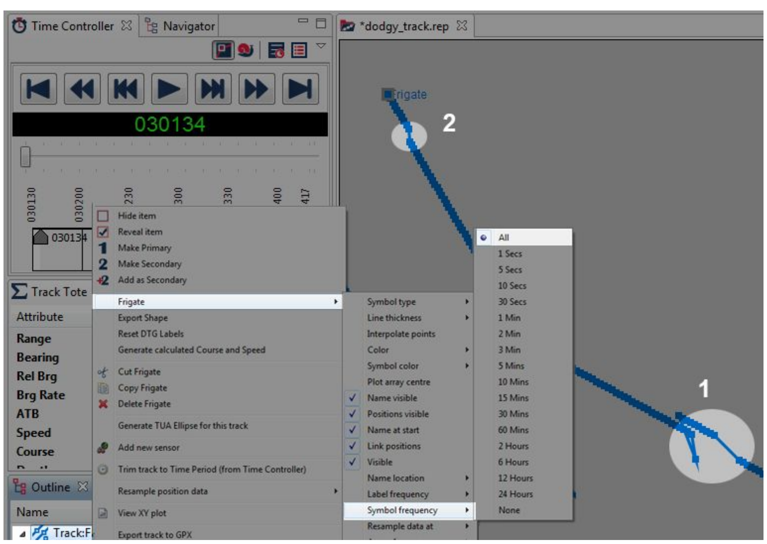
As you can see, we have our blue ownship track called **Frigate**. The vessel is travelling from the North in a general South-East direction. You probably won't be able to see it on your screen, but there is a period of missing data on this track, as well as several jumps in the data itself (this will become obvious in the next step).

Note: jumps often occur when an inertial navigation system receives an external update. We'll fix these first.

Understanding the data

Now we'll make the track and our data easier to understand by making the track data points visible:

1. Open the **Outline** view and you'll see **Track:Frigate**.
2. Left-click on the drop-down arrow (to view the contents of the track) and you will see **Positions (156 items)**. This is the data contained within the track.
3. Now, right-click on **Track:Frigate** and a popup menu will appear.
4. Click on **Frigate > Symbol frequency > All**. The track will now show a symbol at every data point. You can clearly see the missing data points about 2/3rds of the way along the track, and soon after the track start (marked 1 and 2, respectively).



5. To better see and understand what is happening here, we'll turn the time labels on. Click on **Frigate > Label Frequency > All** and you will then see the time labels for each point
6. Now have a look at the times around each block of missing data. The jumps around '**1**' have consecutive time steps, so they're clearly jumps. But the jump

around ‘**2**’ shows a period of about 9 minutes of missing data - we’ll fill that missing period in with data from another source.

We will now fix those gaps.

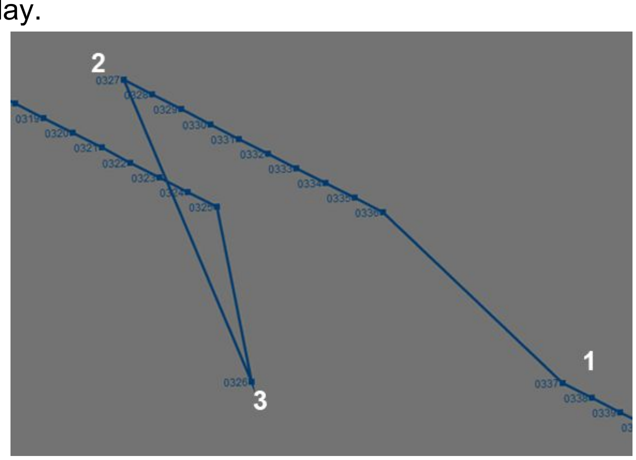
Splitting and fixing the first gap

Zoom into the large gap about 2/3rds of the way along the track (where it looks like almost a 90° starboard turn is made, marked ‘**1**’ on the image above).

Though this part of the track looks like 3 sharp turns were made, it’s actually three jumps in the data. We will now go through and fix these, and will start by clearing the last jump.

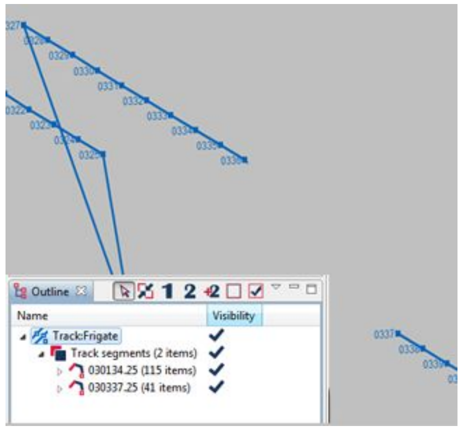
Here the vehicle is travelling in an ESE direction, so:

1. Right-click on the first valid point in the southernmost leg of data (it's the first point in the last leg of straight line data, marked ‘**1**’ on the image below) and a popup menu will display.



2. Select **Split track before 0337** and the track will split into 2 pieces.

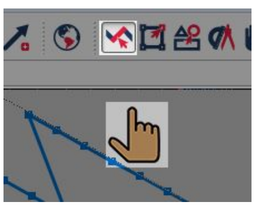
3. Look at the **Outline** view and you will see that **Track:Frigate** now contains **Track Segments (2 items)**.



We now need to align the tracks. First, we'll hide the labels as they do clutter the display somewhat. In the plot editor, right click on the **Frigate** track line then select **Frigate > Label Frequency > None**. The labels will be hidden.

Align the tracks

To align the tracks, we need to drag one track segment to the other.



1. Click on the **Drag Track Segment** button on the toolbar (or press **Alt+1**). The **Bearing Residuals** view opens and, as we're now in drag mode, the mouse cursor will change to a brown hand.

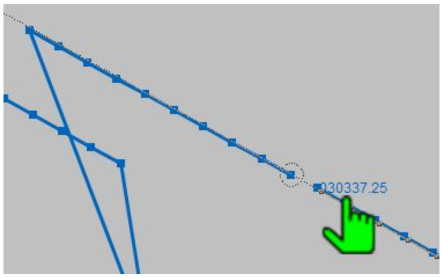
2. For this step, we need to drag the entire track segment, so click on the **Translate mode** button near the top of the **Bearing Residuals** view.

3. Now, move the mouse-cursor over the first point of the lower segment of track, labelled **030337.25**. It will turn green when it is in the correct position.

4. Start dragging the track upwards and leftwards over the dangling end of the other track segment.

   As you drag the track, you will see that extension legs are plotted on each end of the track, with a marker circle plotted at the distance along the extension equal to the distance from the end point to the next first point on the track.

6. Now fine-tune your drag operation to put the 'target' over the last point on the blue track, then release the mouse-button to drop the track.



Merge track segments

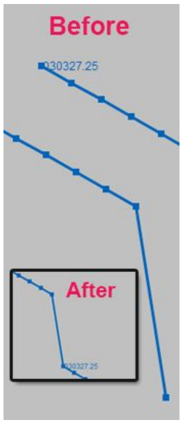
We will now merge the track segments:

1. In the **Outline** view, right-click on **Track segments (2 items)**, a popup will appear. ❑

2. Click on **Merge all track segments**. The tracks will combine and **Track:Frigate** will once again show **Positions (156 items)**. ❑

Split and fix the second jump

We'll now correct the jump to the north of the one we just fixed.



1. Open the **Time Controller** view and move the time slider to **030326**, this is where it looks like the vessel is travelling west of north and then does a sharp turn to the south-east (marked as position 2 on the earlier image). ❑
2. Right-click on the track data point and select **Split track before 0327** and the track will split into 2 again. ❑
3. Now, click on the **Drag Track Segment** button again (or press **Alt+1,** if you like to learn the shortcut keys) and drag this end of the track down and to the left and position it on the dodgy point at the end of the good track section. ❑
4. In the **Outline** view, combine the tracks again. ❑

Split and fix the third jump

We'll now do the same with the third jump:

1. Right-click on the data point at **0326** (if you hover your mouse cursor over the data point, a tooltip will appear giving details at that point, marked as position 3 on the earlier image) and select **Split track before 0326**. ❑

2. As before, drag this track to the good section and merge the tracks into 1. ❑

Add third party data

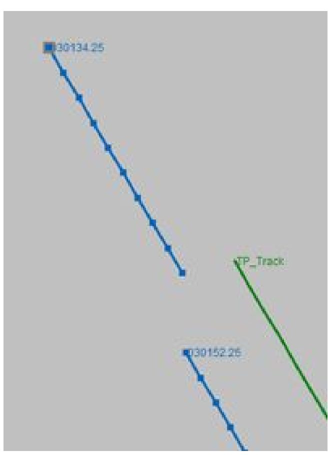
We have now removed the jumps from the track. But, as we established earlier, there is still a missing period of data near the start of the track. Fortunately we have been able to obtain data for this period from a slightly less accurate typed log. We will now add this external data to fill the remaining gap.

1. In the **Navigator** view, locate the file **“third_party_track.rep”**. ❑
2. Drag and drop the file into the plot editor and select **Over The Ground (OTG)** when the track mode appears. ❑
3. Click **Ok** and you will see the new track appear to the north of **Track:Frigate** in the plot editor and appear as **Track:TP_Track** in the **Outline** view. ❑

Now we need to position and merge this track with the blue track.

Position the new data

To make it easier to merge the tracks, we need to move the new track closer to the Frigate track.



1. Click on the **Drag whole feature** toolbar button (**Alt+3**), move it over the **Track:TP_Track** until the mouse cursor changes green, and then drag and drop the track just below the ‘**g**’ in the **Frigate** track title. ❑
2. Zoom in on the gap in the **blue** track. ❑
3. Right-click on the data point to the north of the track, and select **Split track after 0143**. ❑
4. Click on the **Drag Whole Feature** button (**Alt+3**) and then drag and drop the top point of the green track onto the southern end of the blue track. Zoom in if need be. ❑
5. Now, click on the **Drag Track Segment** button on the main toolbar (**Alt+1**), and drag and drop the southern end of the green track onto the northern end of the blue track. ❑

Now that our tracks are lined up, we need to merge them.

Join track segments

To join the track segments:

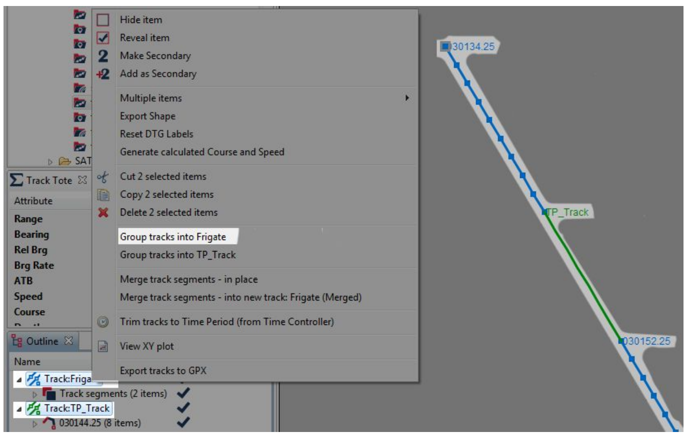
1. In the **Outline** view, **Ctrl+click** to select both tracks.

2. Right-click and then select **Group tracks into Frigate**. Now, only the **Track:Frigate** remains in the **Outline** view.

3. If you click on the drop-down arrow next to **Track:Frigate** you will see **Track segments (3 items)**. However, we don't want a segmented track, we want a complete track.

4. Right-click and select **Merge all track segments**. You will now see **Track:Frigate** which contains **Positions (164 items)**.

   Our track is now complete.



In this stage of the tutorial, you fixed some jumps in the ownship track, and filled in a period of missing data using external data.

You can close the current plot now, as you won't need it for the next stage. No need to save changes.

Groom sensor data

This tutorial is going to work through creating and preparing the ownship sensor data ready for production of **Target Motion Analysis (TMA) tracks**.

Load datafiles

For this tutorial we will be using a new set of datafiles. In the **Navigator** view:

1. Go to the **sample_data > S2R** folder.

2. Double-click on the **"nonsuch.rep"** file. The **Select track mode** dialog will display.

3. Select **Dead Reckoning (DR)**... from the dialog and click on **OK**. The ownship track will load into the plot (**Track:NONSUCH** in the **Outline** view). ❑

4. Click on the **Fit to Window** button. ❑

5. Next, from the **Navigator** again, drag in the **"sensor.dsf"** file. ❑

6. The **Import Sensor Data** dialog will display. ❑

7. You can go through the tabs by clicking on the **Next** button; but for now, you can also click on **Finish** to select the default options. ❑

Note: the plot area can quickly become cluttered when showing bearing lines. So, by default, you will not see the sensor data loaded from **sensor.dsf** file on the plot.

Make sensor data visible

Though that is the default option, we now wish to show that sensor data we've just imported. So, in the **Outline** view:

1. Click on the down-arrow next to **Track:NONSUCH**. ❑

2. Click on **Sensors (1 items)**, to expand it. ❑

3. Then click on **Sensor:Sensor_A (52 items)** to select it. ❑

4. In the **Outline** view toolbar, click on the tick button (**Reveal selected items**), and all the sensor data will be revealed (we call these **bearing fans**). ❑

Resolve ambiguity

In our data you can see one bearing fan heading off to the WNW, and the other off towards the NE—this is typical of a towed sensor as they produce both a true bearing to the target, and a mirror-reflection, ambiguous bearing.

Drop ambiguous data

Intelligence tells us that the actual contact is off to the NW, so we can dismiss (drop) the ambiguous data set of bearings that generally point NE:

1. In the **Outline** view, expand the **SENSOR_A** dataset.
2. Select all of the child items by clicking on the top value, scroll to the bottom of the list and then press and hold the **Shift** key on your keyboard while clicking on the bottom value - this will select all 52 sensor items.
3. Right-click on any of the entries and select **Keep starboard bearing**; the port bearings will disappear from view.

Next, we need to edit the raw sensor data.

Open grid editor

One method of editing the raw sensor data is by using the **Grid Editor**.

1. Click on **Window > Show view > Grid Editor**.
2. The **Grid Editor** view will display, but it will be blank until we indicate the data to edit.

Indicate data to edit

1. The grid editor 'listens out' for the current selection on the **Outline** view. However, if the selected items in the **Outline** view are not suitable for editing in the grid format, nothing will be seen, e.g. as we have just seen, after hiding the port bearings in this tutorial and opening the grid editor, the grid is empty.
2. To populate the grid area: click on the **Track Segment 220350.04 (826 items)**, which is just under the **Track:NONSUCH** in the **Outline** view. The grid edit view will now populate.

   Note: this behaviour can be cancelled by clicking on the **lock icon** button in the **grid editor** toolbar. If you open the **Track:NONSUCH** item and select the positions you'll see them appear in the grid editor.
3. Now click on the sensor data for **Track:NONSUCH > Sensor:SENSOR_A (52 items)** - the data from this track will now populate the grid editor.

Tidy your interface

As mentioned previously, the interface can become easily cluttered. However, if you have a dual monitor and need all the windows and views open, then you can drag-and-drop views onto your other screen by clicking and holding the individual view tabs, and then moving them or resizing them as required.

View grid data

The scrollbar on the right-hand side lets you move forwards and backwards through the data (the newest items are shown at the top). Most cells in the grid are editable, including the date, and blue and red buttons are provided in the toolbar to add or remove rows. Clicking on the blue **Add** button will insert a duplicate of the currently selected row immediately beneath it—a major time-saver compared with manually entering data.

Work on an attribute

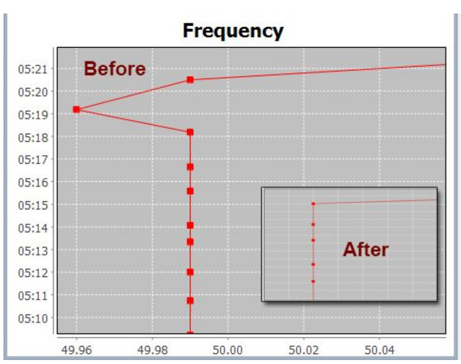
In addition to straight-forward text-editing of data, selecting an attribute offers further editing capabilities. For example:

1. In the **Frequency** column, click on the header cell itself (where it says **Frequency**). you'll see a graph appear in the bottom half of the view. This graph is a ‘waterfall’ display of frequency, with the most recent value at the top.
2. **Zoom in** on data by dragging your mouse on the graph using a top-left to bottom-right motion.
3. **Zoom out** by dragging bottom-right to top-left.

Fix dodgy frequency observation

If you zoom out to look at all the frequency data you'll see that whilst this data seems fairly constant near the top of the dataset (along the **49.99 frequency value**), there are occasions where the data value seems too low (when viewed in the context of a steady ownship track).

As such, we're going to fix an errant data point by dragging it into a better position.



1. Zoom in on the data around the time **05:20.** You will see that the data-point at **05:19:11** is quite a lot lower than its neighbours.
2. Move this data point by clicking inside the square data point and dragging the symbol to align it with it's neighbours. Remember, you can zoom in multiple times for greater precision.

Smooth period of data

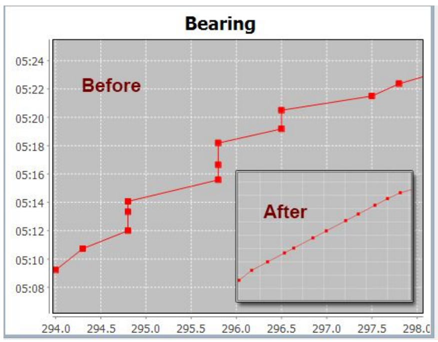
To smooth a block of data, we will switch to bearing data:

1. Click on the **Bearing** column, the graph of bearing data will appear. ❑

2. Around the period **05:10** to **05:20** on the graph, you will see three step-ups in the data, zoom in on these three steps. ❑

It appears that the last smooth data point before the steps is at **05:10:44**, and the first after the steps is at **05:21:30.** We're going to interpolate the data points between these two values.

To do so, we must inform Debrief which points we intend to keep:



1. In the grid (the top half of the editor) click on the empty space to the left of row **05:10:44**. ❑
2. Now, hold down the **Ctrl** key and click on row **05:21:30**. As soon as you do this, the **Interpolate** (**calculator**) button in the grid editor toolbar will be enabled. ❑
3. Click on the **Interpolate** button and the bearings for the selected data points will be smoothed along a straight line (as shown in the image to the left). ❑

Note: you can select **Undo** from the edit menu, or press **Ctrl+Z** on your keyboard, to undo an interpolation operation. ❑

Getting clever with interpolation

Remember, in the previous step, we only selected a single point at each end of what could be considered poor data. However, there are two other more advanced ways of doing an interpolation.

1. If we had selected multiple points at the ends of the dodgy data then Debrief would have made the interpolated values fit through a cubic spline that passed through the selected points.

2. If we had selected one or more points in the middle of the dodgy data (in addition to point(s) at the ends) then Debrief would have fitted a curve through the end and midpoints of the dataset.

Now that we've tidied the sensor data, we can move on.

Set array offset

As you will see, the sensor data is ambiguous; this can happen when the data has been produced by a towed array.

In this current scenario, Debrief is plotting the sensor cuts against the attack datum of the platform, but we need to apply an offset of 451m in this particular example for this sensor (this is -451m). To do this:

1. Select **Sensor:SENSOR_A** in the **Outline** view. ❏
2. Open the **Properties** view and you will see the **SensorOffset** attribute. Enter **-451** in this field, then press **Enter**. As you do so, the bearings on the plot will adjust to reflect this change. ❏
3. To see where the current array centre is, in the **Outline** view, click on **Track:NONSUCH**. ❏
4. Then, in the **Properties** view, under **Format**, you will see **PlotArrayCentre**, change it to **Yes**. A cross will appear astern of the current submarine location. ❏

Note: we will be using the track and sensor data from this tutorial in the next tutorial, so ensure you save it if you're going to take a break and move onto something else.

OPTIONAL — Reduce data density

Though it isn't necessary for this particular scenario, Debrief does allow you to reduce the density of sensor data:

1. In the **Outline** view, right-click on **Sensor:SENSOR_A**. ❏
2. In the drop-down menu, click on **SENSOR_A > Visible Frequency**, then select the filter required. ❏

Adjusting these options will filter out sensor data and hide sensor observations between the specified frequency. The sensor cuts are not deleted, just hidden; and can be made visible again by selecting a smaller, visible frequency.

Note: a more permanent way of doing this is to right-click on the parent track, **Track:NONSUCH**, select **Resample position data at**, and select the required period.

We've now got a track with lovely smooth sensor data.

Save your data

As mentioned earlier, this sensor data and track will be required for the next tutorial; so, if you don't intend to jump straight onto it, ensure you save this file:

1. Click on **File > Save**. ❏
2. You have to save the file with the **dpf** extension, and also **select the folder** where you will save the file. ❏
3. Enter the **filename**, e.g. **nonsuch_plus_data.dpf**. ❏

4. Click on **Ok**.

Generate a target track

This tutorial guides you through creating a target track segment from bearing data to represent a period of straight-line motion; this can be interpreted as a **Manual Target Motion Analysis (TMA) Solution**.

1. We will be using the track and sensor data from the previous example, so if you closed it, reopen it now.

2. In this tutorial, we will be concentrating on the plot, so you can close the **Grid Editor** view.

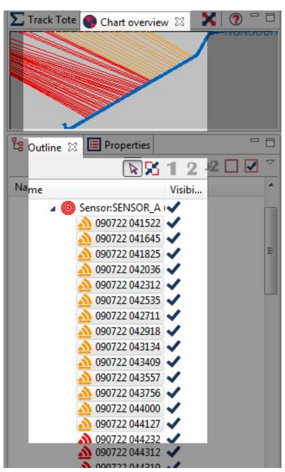
Deciding the contact period

A capable analyst will be able to recognise a couple of periods of steady bearing rate that lend themselves to being the basis of TMA solutions.

The first period of cuts we're going to be using here are the first 15 or so sensor cuts. We can see that at the end of this period, there are 2 bearing lines very close together and then a gap, followed by a small group of 3 bearing lines, and then another gap. After this second gap, there is then a period of about 10 unsteady cuts, and then another steady period; we'll be using that for a solution a little later. Experience shows us that, since Ownship is steady throughout this period, these bearing patterns indicate that the subject vessel was first on a straight course, then it manoeuvred, then it assumed a second straight course.

Highlight contacts

We will change the color of the contacts to make our first solution easier to see and work on:



1. Identify the last bearing line you will use for the first period of steady bearing data.
2. Double-click somewhere on that bearing line and it will then be highlighted in the **Outline** view (I selected the data point named **090722 044127**).
3. Click on that item in the **Outline** view to select it.
4. Now, scroll back up to the start of the sensor data, hold down the **Shift** key, and then click on the first entry—this period of sensor data is now selected.

5. To change the colour, right-click anywhere on the selected group and select **Multiple items > Color > Orange**.

6. The bearing lines in the plot area and the selected items in the **Outline** view will turn orange.

Generate the TMA segment

Next, we're going to create a solution from this selected (orange) data. When we generate a TMA segment based on sensor data, Debrief creates a track segment of steady course/speed data points, with one data-point at the time of each sensor cut used to generate the segment. Debrief has a **Generate TMA Wizard** to help you with this.

With the set of data still selected in the **Outline** view (or reselect it if you've lost the selection):

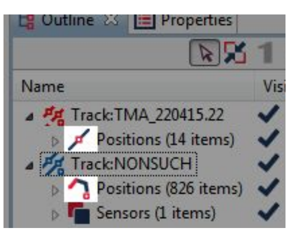
1. Right-click on the selected sensor data and select **Generate TMA solution from selected cuts**. The **Generate TMA Segment** dialog will show, asking you to specify the offset to the track start.
2. Enter a range of **1 nautical mile (nm)**, and you can leave the bearing of **300.8** as is (it's using the bearing from our first sensor cut).
3. Click **Next** and you will see the initial solution dialog.
4. Based on our understanding and analysis of the sensor fan in the plot area, enter **220** for the initial estimate of the course and **6 kts** for the speed estimate.
5. Click on **Finish**.

A red track will appear on the plot, labelled with the TMA solution and the time of the first cut used (**TMA_220415.22** on my plot—all solutions have times in their name to make them easier to manage).

Recognise track data

We will now look at the track data for this TMA solution in the **Outline** view.

1. First, expand the new track (mine is called **Track:TMA_220415.22**) in the **Outline** view and you will see the **Positions (14 items)** child item.



2. If you look at the icon for this particular child item, you will see that it shows a straight-line section of data with a tiny red compass rose on it. However, if you also look at the equivalent object for **Track:NONSUCH**, you can see that it shows a non-straight-line set of positions (with 2 turns). Also, note that on the plot, the name of the **Track:NONSUCH** TMA segment is shown in italics, to denote that it's not based on actual position recordings.

Put the tracks on the tote

To view residuals, we must indicate to Debrief which is the ownship and which is the target. Debrief can hold many more than two tracks, but we must inform it which two tracks we want to compare.

1. In the **Outline** view, select **Track:NONSUCH**. ❏
2. In the **Outline** view toolbar, click on the **1 (primary)** button. ❏
3. Now, click on the **Track_xxxx** item (as mentioned, mine is **Track:TMA_220415.22**). ❏
4. Click on the **2 (secondary)** button on the toolbar. ❏
5. Now, switch to the **Track Tote** view (**Window > Show View > Track Tote** if it is not open) and you'll see it contains both our primary and our secondary tracks. ❏

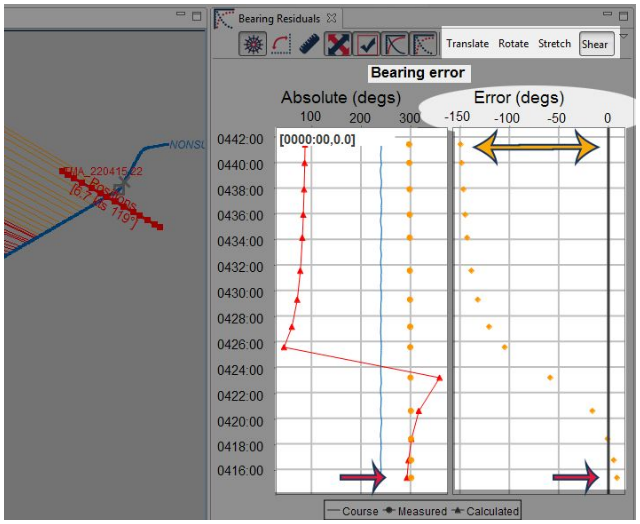
Debrief now knows what tracks we wish to compare, so we will manipulate the track segment to minimise bearing and frequency residuals. But first, let's look at the **Bearing Residuals** view.

The Bearing Residuals View

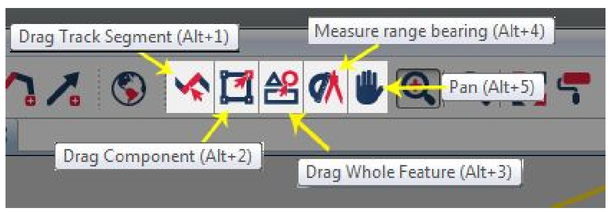
To open the **Bearing Residuals** view, click on the **Drag Track Segment** button on the Debrief toolbar (**Alt+1**). The **Bearing Residuals** view will display. ❏

Note: if the **Bearing Residuals** view is small, make it larger for this next exercise. As with all views, you can resize or move them by moving your mouse over the sides of a view and, when the cursor changes to a multi-directional arrow, drag in any of the directions indicated; or, if required, by clicking on the view's tab, you can then either drag the view to another part of the display or to another connected monitor.

Looking at the **Bearing Residuals** view, you will see that it's titled **Bearing Error**. The aim with this view is for the analyst to reduce the residual error—the error between the measured and the calculated bearing—by manipulating the track **manually in the plot editor**. As this is carried out, a real-time view of the track manipulation is displayed in the **Bearing Residuals** view.



This manipulation is carried out using one of the **translate**, **rotate**, **stretch**, and **shear** modes, which are located on the **Bearing Residual** view toolbar, in conjunction with Debrief's **Drag Track** functions, which are located on the **main toolbar**.



When you have selected your drag mode and started to manipulate the active track in the plot editor, the resultant effects will be seen in both panes in the **Bearing Residuals** view: **Absolute (degs)** and **Error (degs)**. The aim here is to drag/position the active track to reduce the calculated error to a minimum, i.e. the **Error (degs)** right-hand pane shows an error of **0 degrees**.

This, in conjunction with the analyst's knowledge, experience, and other available data, such as narratives, will enable them to perform an accurate scenario assessment. Let's do this now.

Manipulate the track segment

1. In the **Bearing Residuals** view, **Absolute** (**degs**) pane, you can see 3 lines: **Course** (this is your vessel's course), **Measured** (the bearing recorded by the sonar), and **Calculated** (the calculated bearing between each ownship position and each position of the hypothetical subject vessel).

   Note: if you don't seen any lines here, click the ship's wheel toolbar icon twice in the **Bearing Residuals** view to switch the plotting of **Ownship Course** off then back on**. The Bearing Residuals** view will now update to show the measured/calculated data.

   The **residual Error (degs)** pane of the **Bearing Residuals** view at the start time of 04:15:22 (obtained from the grid editor data), shows a residual error of just under 10° (you can zoom in by clicking and dragging top-left to bottom-right in the pane). We need to reduce this as much as possible, so must select the drag operation most suitable. ❑

2. There are 4 drag mode buttons in the **Bearing Residuals** view, these control how you drag the track. They are labelled, and have functions, as follows:

    ○ **Translate -** you can drag either end of the track: this changes range and bearing from the source, but maintains target course and speed.

    ○ **Rotate -** here you can drag either end of the track: doing so maintains the target speed but changes the target course.

    You can also drag the track from the middle in this mode, but when **rotate**, **stretch**, or **shear** mode is selected, a **Translate** operation is performed (the button in the **Bearing Residuals** view doesn't switch to Translate though).

    ○ **Stretch -** here you can also drag either end or the middle of the track: stretch allows you to maintain the target course but change the target speed. If the centre-point of the track is dragged, the track moves in and out adjusting as necessary to adhere to the start/end points on the host platform bearing fan.

    ○ **Shear -** you can drag either end or the middle of the track: this option allows you to change both target course and speed.

    Note: whilst the **Translate** and **Rotate** operations are available for all types of track segment, only the **Stretch** and **Shear** operations are suitable for application to **straight-line TMA Segments** (the sums just get too complex when applying these operations to real track data). To help you, the hand cursor will only turn green over straight-line TMA Segment hot-spots.

    As mentioned earlier, though you select the above drag options in the **Bearing Residuals** view, these selections are used in conjunction with the following drag buttons from the main toolbar:

    ○ **Stretch**, **Shear**, and **Rotate** are used with the **Drag Track Segmen**t function.

    ○ **Translate** can be used with **Drag Track Segment**, **Drag Component**, or **Drag Whole Feature**.

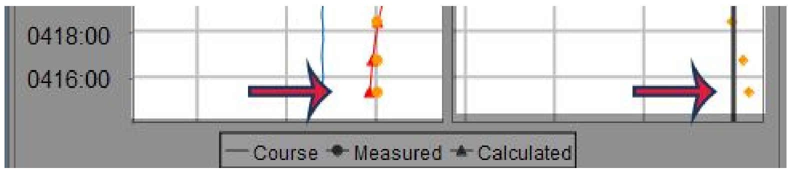
    By default, the mouse cursor hand is brown. When it is this colour, you cannot modify the track from this particular point—it isn't a *hot-spot*. However, if you move the cursor over a point on the track which does allow you to manipulate (or move) that point—a *hot-spot*—the cursor will change colour to green. As mentioned in the bullet points above, this will either be at the end, and/or in the middle of a track.

3. Now, select **Shear** mode and then click on **Drag Track Segment** on the main toolbar (or press **Alt+1**) and move your cursor to the start point of **TMA_220415.22**. ❑

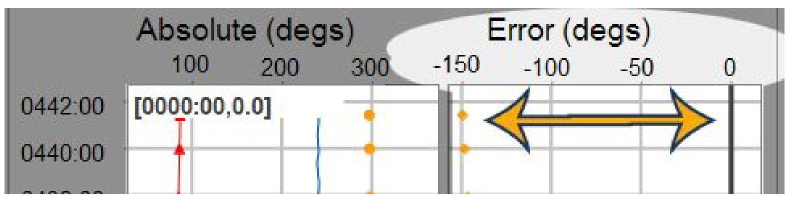
4. When you are in the position to drag the track point, the mouse cursor will turn green, you can now drag. ❑

Bear in mind, the data points we have must correspond to the bearing fans, so the objective here is to both reduce the residual AND position the start of the track on the bearing line. You will probably need to zoom in on the track and in the **Error (degs)** pane.

5. Try and reduce the error to as close to **0°** as possible by positioning the track start point on the first bearing line (I obtained a residual error of **1.8°** when I dragged the start point to **05° 00.846' N 029° 56.912' E**).



6. Do the same with the end of the track. As you can see from the image below, there is approximately a **150°** error as I start my manipulation (in the end, II obtained a residual error of **-0.02°** at position **04°58.778' N 029 58.585' E**).



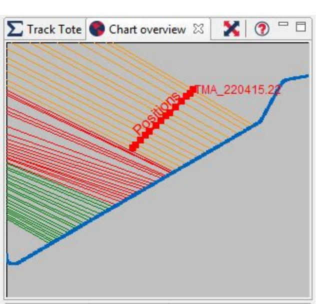
Now we'll look at expanding on this with another TMA segment.

Generate second TMA segment

Looking at the sensor data, it would appear that the turn is represented by about 14 cuts before a further straight line section of data.

As you look at the fan of sensor data, you can see there's an early block of bearing lines that are roughly parallel. Then, towards the end of the track, the bearing lines appear to converge steadily around a single point. In between these two periods of steady data, the lines jump around a little, and this represents the period where the target vessel is changing course and/or speed.

Now, let's work with the plot again…



1. Click on the first sensor cut of the second steady set of bearings - that cut will be highlighted in the **Outline** view (I selected **090722 050326**).

2. Now scroll down to the end of the data and **Shift-click** the last sensor cut. We've now selected the cuts to be used for the second track segment.

3. Right-click on one of the selected items and select **Multiple items > Color > Green**. The cuts will turn green. ❏

4. Right-click again on the selected cuts and select **Generate TMA solution from selected cuts**, and complete the wizard once again: ❏
   a. Enter a range of **1 nautical mile (nm)** and you can leave the bearing of **292.3** as-is (it's using the bearing from our first sensor cut in this selection, which represents as good a start point as any). ❏
   b. Click **Next** and you will see the initial solution dialog. ❏
   c. Based on our understanding and analysis of the sensor fan in the plot area, enter **220** for the initial estimate of the course (but it won't be, because the earlier spacing in the bearing fan shows us that the vessel has turned) and **6 kts** for the speed estimate. ❏
   d. Click on **Finish** and the second track will be plotted. ❏

As mentioned, due to the earlier turn, the new track is probably wrong; but we can manipulate the track and see what solution we can arrive at.

Refine second solution

Now, locate this new solution in the **Outline** view:

1. Click on the track to select it (mine is called **Track:TMA_220503.26**). ❏

2. Then click on the **2 (secondary)** button on the toolbar to designate it as the secondary track. ❏

   As soon as you mark it as secondary, you will see it get listed in the **Track Tote** view. ❏
3. Now, start manipulating and refining this solution (I obtained the following for the track start point: a residual error of **-2E -5°** at position **04° 57.701' N 029° 57.098' E**; and, at the end point, a residual error of **0°** at position **04° 57.433' N 029° 54.689' E**). ❏

Merging one or more tracks

Select and merge tracks

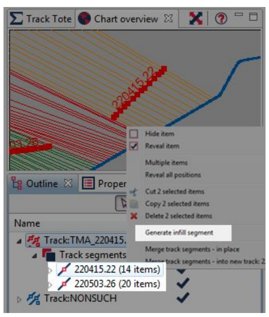
In the previous tutorial we generated two track sections and, though they are separate tracks on our plot, we know that they belong to the same vessel; so, we will now combine them into a single track.

1. Using the **Outline** view, **Ctrl+click** to select both tracks.
2. Right-click on one of the selected tracks and select **Group tracks into TMA_xyz** (it doesn't matter which one you select).
3. The tracks will merge into 1 (the 'other' track will disappear). However, there is still that large 'turn gap' remaining. We will now use Debrief to *infill*, or bridge, that gap.

Provide infill positions

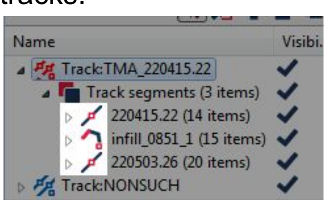
Debrief is able to link track sections by infilling the gap between them.

1. Click to expand **Track:TMA_xyz** and you will see it contains **Track Segments (2 items)**.
2. Expand this and you will see 2 tracks with different numbers of items (in mine, I have **220415.22 (14 items)** and **220503.26 (20 items)**).



3. Select both tracks (**Ctrl+click**)
4. Right-click on either one and select **Generate infill segment**.
5. You'll see a new segment appear both in the **Outline** view and in the **plot**.

Notice that the infill segment is shown as a dotted line. This indicates that this track segment is not based on any real bearing data, it's just been calculated to join both tracks.



Also, if you look in the **Outline** view at the **Track Segments**, you will see the 3 individual tracks listed. Take note of the icons: 2 are straight line tracks, and the other shows a turn.

Merge tracks

You can continue to keep your target track represented as 3 or more track segments for as long as you like. But, when you need to export the data for presentation or subsequent analysis then you must merge them:

1. Select the parent item for the track segments (**Track segments (3 items)**).

2. Right-click on it and select **Merge all track segments**. The tracks will merge. ❑

Conclusion

That's the Single-sided reconstruction tutorial using Debrief finished. I hope you can see how easy it is to use Debrief, as well as appreciate the power and potential for working with your data. The next tutorial takes us to the next step and walks us through a Semi-automatic Target Motion Analysis scenario.

Signed: ________________ Date: ___________________

Name: ____________________ Date: ________________

Please forward feedback on this tutorial to **support@debrief.info**

Semi-automatic TMA walkthrough

Semi Automatic Track Construction (SATC) was developed in response to the challenges of handling and solving multi-leg target tracks using Debrief's manual TMA algorithms. SATC provides an automated means of obtaining an optimal solution based upon disparate "packages" of analyst knowledge, measured data, and logical/geometric deductions (collectively these are called “contributions”). This tutorial looks at both single and multi leg TMA solutions.

Single Leg Solution

The first tutorial in this set will involve an engagement where there is just a single leg of target data. Here we follow 5 steps: 1) find the sample data, 2) load a simple scenario, 3) create a scenario, 4) generate a solution, and 5) convert the solution to a track.

Finding the Sample Data

There are two ways in which we can load sample data. We will be modifying the sample data, so ensure you’re not working on data that is shared with other users (the master data).

The first thing we need to know is whether your Debrief installation folder is on your local machine, or on a Network Shared Folder:

- If it’s on a **local machine** goto **Sample Data if Debrief is on a Local Machine** (in this tutorial).
- If it’s on a **Network Shared Folder**, go to **Sample Data if Debrief is on a Network Shared Folder** (in this tutorial).
- **Sample Data if Debrief is on a Local Machine** (again, in this tutorial).

Debrief NG stores its data in a local folder called a **Project**. While these can be created to store new analysis data, it's also possible to select existing data directories as linked sub-folders.

Note: if the **Create Project** form opened when you started the application, you will have already provided a project name. If you also indicated that you would like sample data to be imported, then you can skip the following steps and move on to the next tutorial (***Multi Leg Solution***).

1. Open the **Navigator** view. ❏
2. Check you have a workspace/parent folder within which your links are stored. ❏

   If you don't have a workspace, refer to the earlier tutorial ***Generating a project***. ❏
3. If you do have one, right-click on the current project in the **Navigator** view. ❏
4. Click on **New > Folder**, and the **New Folder Wizard** will open. We will now link to an existing folder. ❏
5. Click on the **Advanced** button. ❏
6. Then, in **Advanced Options** click on **'Link to alternate location (linked folder)'**. ❏
7. Click on **Browse**. ❏
8. Navigate to your folder, select it, and click on **Finish**. ❏

Note: if you don't have your own data use the **sample_data** folder in the Debrief installation folder (for a deployed installation), or **org.mwc.cmap.combined.feature/root_installs** (for a development version of Debrief).

Data Linked to from Local Machine

Great, that's your data now linked. Now, just repeat this process to add any other data directories you want to load data from.

On your first attempt at doing this it's likely that you've just created a link to the sample data, but you may also wish to create a link to a shared working folder at some point.

Now we can move onto the tutorial. Go to ***Loading a simple scenario***.

Sample Data if Debrief is on a Network Shared Folder

We will now copy in an existing set of sample data from the **Debrief\DebriefNG** Network Shared Folder.

1. Open the **Navigator** view. ❏
2. Define a project for your work. This is a parent folder within which your data (or links to data) is stored. ❏

   If your **Navigator** view is empty, refer to the earlier tutorial, ***Generating a project***.

Now we will drag in our data:

3. Open **"Windows Explorer"**. ❏
4. Arrange your windows so you can see both your sample data folder and Debrief. ❏
5. Drag the **sample data folder** from **"Windows Explorer"** and drop it into Debrief **Navigator**. ❏

A dialog will open asking you if you wish to **Copy files and folders** or **Link to files and folders**, select **Copy**. ❏

Sample Data Copied from Network Folder

Great, that's your data now copied in. Now we can move onto the tutorial.

Loading a simple scenario

To produce target solutions we must load some ownship data. We'll do that now.

Find SATC subfolder and Load Ownship Track

In the **Navigator** view:

1. Open the **SATC** sub-folder and either drag the **"L1_OwnshipTrack.rep"** file into the Debrief plot area or right click and select **Open With > Debrief Plot Editor**. ❏
2. Select **Over the Ground (OTG)** mode. ❏

You will now see the blue ownship track in your window.

Load ownship sensor data

Next, we will load the ownship sensor data (this is the sonar data):

1. Drag and drop the **"L1_OwnshipSensor.dsf"** file onto the plot, the **Import Sensor Wizard** will begin. ❏
2. You can leave this **sensor contact name** as **Plain**. Click on **Next**. ❏
3. Choose a **color** for the sensor data, then click **Next**. ❏
4. Select the **value** check box so that the sensor data will be visible, then click **Next**. ❏
5. In the **Rainbow shades** dialog, select the check box to shade the sensor cuts. Click on **Finish**. ❏

The dialog box will close and the fan of ownship bearing data (sonar data) will be seen radiating north-west and north from the ownship track. If you don't see this, then repeat the above steps (making sure you've loaded the correct files).

Now we've loaded our data we can create a scenario.

Create scenario

Debrief uses the term ***scenario*** to describe the collection of data that is collated in order for SATC to produce an optimal 'solution'. Let's create our first scenario now.

Create scenario based upon cuts

We will use all of the ownship sensor data to create our scenario:

1. Open the **Outline** view. ❏
2. Open **Track:OWNSHIP**. ❏
3. The ownship track has a **Sensors (1 items)** object, open this and you will see the single block of sensor data, titled **Sensor:Plain (11 items)**. ❏
4. Right-click on it and select **SemiAuto TMA > Create new scenario from these bearings**. ❏

Create scenario based upon cuts

A new item with the name **121213.14** has been created in the **Outline** view (the item name is auto-selected from the Date-Time-Group (DTG) of the first item in the bearing data).

You will also see that the **Maintain Contributions** view has opened. A contribution is a piece of information that is used in the development of target solutions. This information can be in the form of a measurement, such as range, bearing or frequency; data; an analyst forecast such as a prediction, i.e. speed range; or, one of a hidden set of analysis contributions that deduce further constraints based on other contributions. This view is used to create and manage your scenario data.

Change scenario name

As nice as it is to try and remember a new set of telephone numbers every time you create a new scenario, it makes sense to rename the scenario to something more meaningful:

1. Select this scenario in the **Outline** view. ❏
2. In the **Properties** view (**Windows > Show View > Properties**, if it isn't visible) change the **Name** value to ***Single Leg***. ❏

Understand the contributions window

In the **Maintain Contributions** view you can see that, in the **Analyst Contributions** section, there is a single **Bearing Measurement - Bearing Data** item listed.

This is the bearing data that will be used to inform all of the subsequent contributions.

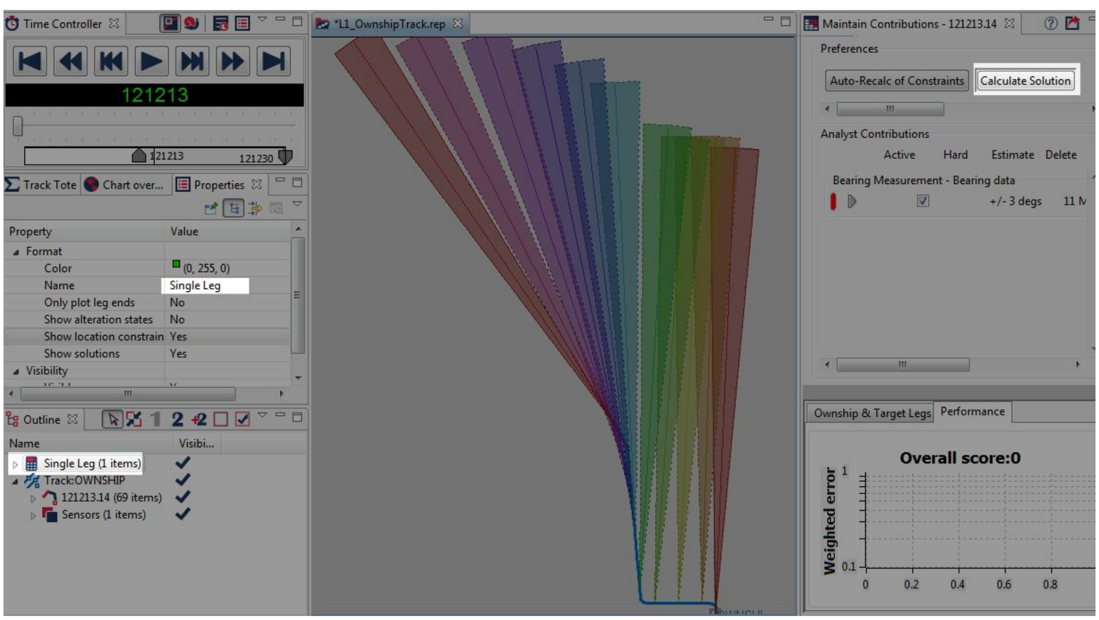
Viewing the bearing measurement contributions

The **Bearing Measurement Contribution** is the set of bearings that represent the 'unknown target track'. You can see that the contribution has an estimated **error** value of **+/- 3 degs**, which means the algorithm will only offer solutions that are within 3 degrees of these bearing measurements.

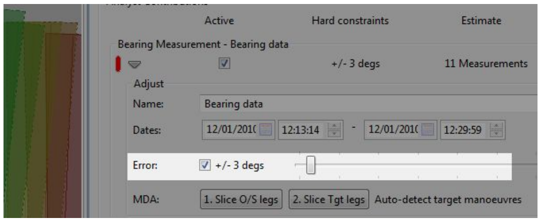
To view these measurements:

1. Select the track **Single Leg** in the **Outline** view.
2. In the **Properties** view, select the **Show location constraints** item.
3. In the **Maintain Contributions** view, click on the **Calculate Solution** button.

A set of "pie-slices" will now appear on the plot, showing the 3 degree allowable error on each bearing measurement.



4. Still in the **Maintain Contributions** view, expand the **Bearing Measurement - Bearing Data** contribution and you will see that it is possible to change the error value on this bearing data by moving the slider. As you move and release the slider you will see the pie slices expand and contract (you may need to click on the **Auto-Recalc of Constraints** button in the **Preferences** area to initiate this).



Note: The **Auto-Recalc of Constraints** is a two-state button that is normally depressed. But, if you have a complex scenario that takes some time to update, you may wish to un-check this button - then the screen will no longer be refreshed as you are interactively adjusting any contributions.

Get ready to specify a target leg

Note: in this sample data, the imported bearings have a maximum range of 12,000 yards (yds). In the absence of a range estimate SATC will restrict them to 30k yds, to stop your PC doing the *Ferranti Reset*.

Also, before we continue, Debrief must be reset to a *predictable state*, so:

1. Restore the **Analyst Contribution** to **3 degrees**.
2. In the **Properties** view, deselect the **Show Location Constraints** (so it shows **No**).

Specify a target leg

It's possible that your deep analysis skills have led you to believe that all of these cuts relate to a single leg of target data: where the target maintains course and speed through the period of interest - this is of great value to the TMA algorithms, since it means the contact must travel through those pie slices in a straight line.

1. In the **Outline** view, select **Track:Ownship > Sensors (1 items)**, and then right-click on **Sensor:Plain (11 items)**.
2. From the drop-down menu, select **Semi Auto TMA > Add to Single Leg > New Straight Leg for period covered by [sensor cuts]**.
3. When the **New contribution** dialog opens up, name this contribution **Leg limits**.
4. Click on **Ok**.

In the **Outline** view, you will see **Single Leg (1 item)** has changed to **Single Leg (2 items)**.

Understand the contributions

So, now there are two contributions for this scenario:

1. a set of bearings through which a solution must travel.

2. an indication that the target will have travelled on a single course and speed throughout the entire engagement.

An observant analyst will have noticed that our two ownship turns have provided a valuable change in bearing rate - exactly what a TMA algorithm needs.

So, now that we've provided the TMA algorithm with some viable data, let's see it calculate a solution.

Generate a solution

Now we have provided background data to SATC, we can generate a solution.

Calculate the solution

Now we're going to generate the solution for this scenario. Before we do this:

1. Click on the **Fit to Window** button to ensure you can see the ownship track and the rough area where the target should be ❏
2. Click on the **Calculate Solution** button. ❏

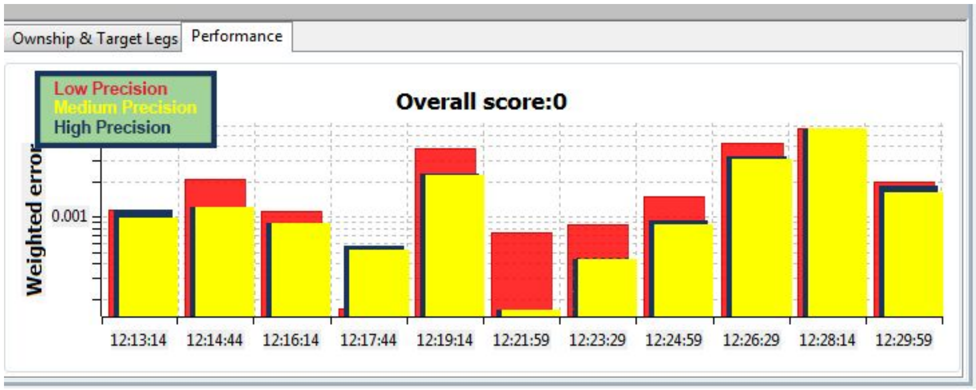
By default, the SATC is set to **Low** precision and uses relaxed constraints. The quite sparse bearing data and ownship maneuvers used in this scenario means that there aren't very many candidate solutions, and an answer will be generated within a second or so. When it has completed its calculations, a new track will appear on the plot.

As you can see, the track will be the same color as you specified for the bearing data in the **Bearing Import Wizard**.

My computer returned a solution of **9.0 kts** on **269°**.

You can alter the precision setting as you require (to medium or high), but you must click on **Calculate Solution** each time to generate it.

As you can see from the image below, which superimposes all 3 precision solutions—**Low** (red), **Middle** (yellow), and **High** (blue)—it is unlikely that you will get a better solution with this set of data; but, you will notice that the **Performance graph** (below the **Maintain Contributions** view) processes more slowly as SATC homes in on a particular solution. You will see that the x-axis shows more cycles have been run through, and that each one moves more slowly. The bars shown in the performance graph represent the sum of the contribution errors at each contact position (state).



Marking your own homework

The data we're using here is from a simulation tool; and this means we have the actual target track to compare against.

2. Drag the file **"L1_SubjectTrack.rep"** into the plot area. ❏
3. Select **Over the Ground (OTG)** as the import mode. ❏

   The track will now display on the plot as a solid red line.

Now, compare the SATC solution with the actual target track. You will see that SATC is actually quite close, with a greater error to the East, near the start. This is because of the low bearing rate near the start; however, both tracks appear to be on a roughly parallel course.

We now have an SATC-generated solution which is very close to the actual target track.

Importing a solution

Debrief has a range of analysis and export capabilities we can use on track objects; however, as our current suggested solution isn't yet a track, we need to import it.

Select scenario

If you've followed the instructions in the previous tutorial correctly then you will have a scenario named **Single Leg (2 items)** in your **Outline** view. Click on it to select it. ❏

Convert to Composite Track

To convert the current solution to a **Composite Track**:

1. Right-click on **Single Leg (2 items)**. ❏
2. Select **Convert to Composite Track (legs)**. ❏

You will see a new track appear on the plot and you will see a new item in the **Outline** view marked as a Track followed by the same name of this scenario (mine is named **Track:Single Leg_0**).

Rename to avoid confusion

To prevent potential problems, we will now rename the imported track:

1. In the **Outline** view, click on **Track:Single Leg_0** to select it.
2. In the **Properties** view, change its name to **Single Leg TMA**.
3. Click away from the **Properties** attribute to change the name.

Tidy the plot

It's very easy for the plot to become cluttered, so we will now use the **Outline** view to hide the red **SUBJECT** track:

1. Click on **Track:Subject** to select it.
2. Right-click and select **Hide Item**.

The track will disappear from view and only the **OWNSHIP** and **Single Leg TMA tracks** are visible.

Manually tune the TMA solution

Now we will adjust this solution in the same way we did for the Single-Sided Reconstruction:

1. Click on the **Drag Track Segment** arrow button in the Debrief toolbar and the **Bearing Residuals** view will open.
2. Mark the **OWNSHIP** track as the **primary track**, and the **Single Leg TMA track** as the **secondary track** (using the mini-toolbar above the **Outline** view).

Now you can drag the **Single Leg TMA** track and see the error residuals move.

Note: the **Absolute (degs)** values graph is easier to read in this instance if you select **Use +/- 180 scale for absolute data** button.

Import as standalone track

Note: the manual-track fine-tuning process steps you have just carried out are more suited to complex scenarios where, because of the lack of contributions available for the system-produced SATC solution, the analyst decides to use the manual TMA tools to groom the track by hand.

However, in this scenario, as the raw SATC did produce a perfectly acceptable solution, we can discard the manual track and import the original TMA solution:

1. In the **Outline** view, right-click on **Track:Single Leg TMA** and select **Delete Single Leg TMA**. ❏

Then, to import our original TMA solution:

2. Right-click on **Single Leg (2 items)** and select **Convert to Standalone Track**. ❏

   You will now see the track has appeared in the **Outline** view as **Track:Single Leg**.

Mark our homework

Currently loaded into Debrief we have both our imported **TMA Solution (Track:Single Leg_0)**, and our **Truth Track (Track:SUBJECT)**; we can now use the application to calculate the distance between these two tracks.

1. Select **Track:Single Leg_0** and **Track:Subject** in the **Outline** view (**Ctrl+click** to multi-select). ❏
2. Right click on either track, select **View XY Plot** and the **View time-variable plo**t dialog box will open. ❏
3. Select **Range**, click on **Ok** and the **Select Primary** dialog will open. ❏
4. Select **Track:Subject** as the **primary track**, and click on **Ok**. The **Subject Range vs Time Plot** graph will display. ❏

This graph shows the start of the track has an error of approximately 130 yds, reducing to 60 near the end. This seems quite good considering the target ranges vary from 8000 to 3000 yards from ownship.

Conclusion

That's the single leg solution complete, so you can now close this Debrief plot. In the next tutorial, we will look at multi leg solutions.

Multi Leg Solution

In this, the more advanced multi leg solution, the scenario has several target zigs, and a number of contributions are required to obtain an optimal solution. Similar to the single leg solution, we need to load the data, then groom it before we can play with the multi-legs.

Loading the data

Load **“L2_Scenario.dpf”** by dragging it from the **Navigator** view into the plot area. ❏

As soon as the plot opens, you will see a blue **Ownship** track, with a dark green bearing fan.

Grooming the data

Now we’ve loaded the data, we now need to groom it. First, we need to understand what our data shows, then we create the scenario, before moving onto zig detection and generating and grooming target legs.

Understanding our data

To understand what is occurring here, use the Debrief **Time Controller** and the **Track Tote** to familiarise yourself with the general motion of the **OWNSHIP** track. As you do, you will see that the vessel starts in the North-East of the plot, then travels quite slowly at 2.5 knots, with two straight legs.

Create the scenario

The most significant block of information that Debrief requires to generate a solution is the bearing data—we need to mark this information as such.

In this scenario, we will use all of the **Ownship** sensor data:

1. Left-click on the drop-down arrow next to **Track:Ownship**. ❏
2. Left-click on **Sensors** to expand it. ❏
3. Select **Sensor:Plain (145 items)**, right-click and select **Semi Auto TMA > Create new scenario from these bearings**. ❏

As with the previous tutorial, you will see the new scenario (called **121200.00**) appear in the **Outline** view, and the **Maintain Contributions** view will open, with just one contribution.

Intro to Zig Detection

Though **SATC's Manoeuvre Detection Aid** is capable of detecting target zigs, the algorithm can only reliably detect target zigs during an ownship straight leg (the geometry is just too chaotic during an ownship turn), so the actual process involves two steps:

1. Detecting ownship legs.
2. Detecting target zigs during those straight legs.

Despite this, the algorithm does make a determined attempt to identify a target zig somewhere during an ownship turn by forecasting the bearing rate in the second leg. If the **post-turn bearing rate** doesn't match that produced in a passive ranging calculation then a turn is assumed. However, because the algorithm won't know when in the ownship turn the target turn happened, it interprets the whole ownship turn as the period of the target turn.

Generate the target legs

As mentioned in the previous step, the process begins with determining the ownship legs. As analysts, we can see what is happening on screen, but Debrief cannot; and, the main information source to enable Debrief to detect target zigs and generate target legs is the bearing data.

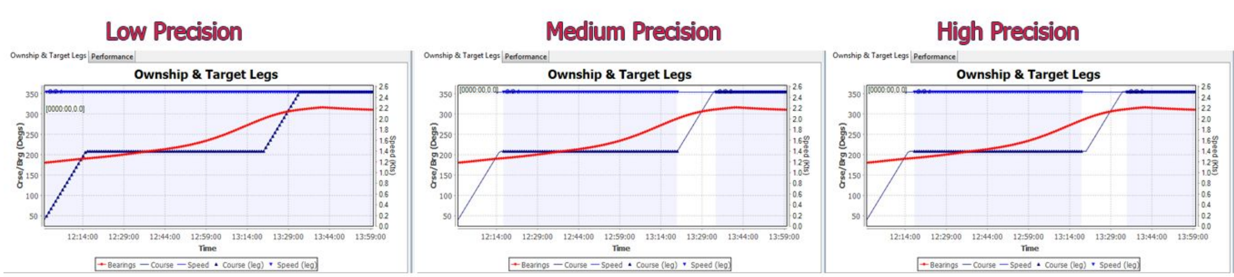
1. In the **Maintain Contributions** view, expand the **Bearing Measurement - Bearing data** section. Near the bottom of the control you'll see the **MDA (Manoeuvre Detection Algorithm)** section.
2. Click on **1. Slice O/S legs** to slice the ownship legs.
3. Now, take a look at the **Ownship & Target Legs** graph at the bottom of the screen to judge the effectiveness of the slices.

   When SATC slices the legs it shades each leg in blue, with transparent gaps between the legs. Ownship legs are labelled with text labels at the bottom of the chart.

   For this dataset, when run with **Low** precision the algorithm thinks there is just one ownship leg.

   However, based on the displayed graph of ownship course and speed against time, this is clearly incorrect; it's just that the ownship turns and speed changes are too subtle for the turn detector.
4. So, change the **SATC precision value** to **Medium** or **High** using the drop-down list at the top of the **Maintain Contributions** view.
5. Click on **1. Slice O/S legs** again.

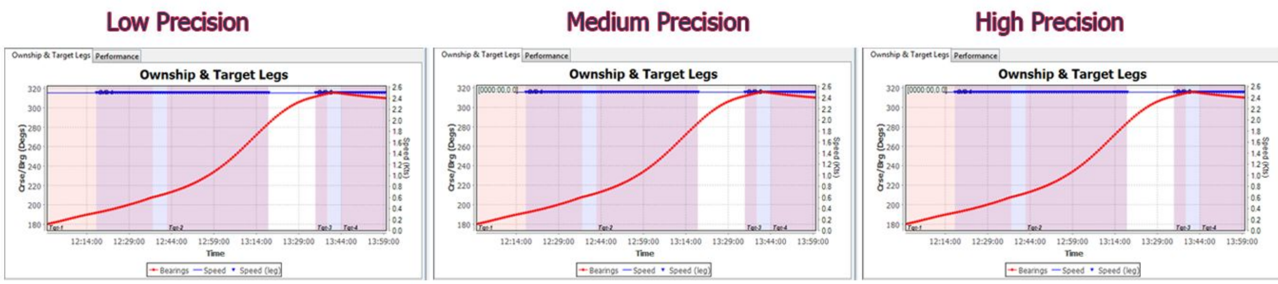
   As you can see from the image below, which compares the 3, you will probably find that **Medium** precision gives the best result.



Once you are happy with the ownship legs:

6. Click on **2. Slice Tgt legs** and the MDA algorithm will produce a series of legs for you.

   As can be seen in the comparison output below, when the precision on my pc is set to medium the algorithm produces four target legs. The algorithm-spotted target turns part-way through ownship legs 1 and 2, and it also assumes a target turn somewhere during the ownship turn from leg 1 to leg 2.



Groom the target legs

Let's have a look at what the algorithm produces from these target legs:

1. Click on **Calculate Solution**. The algorithm will produce a target track and show it on the plot.

   Looking at this target track, you should see that the very short target leg 3 is quite similar to leg 2. Switching back to the **Ownship & Target Legs** graph, you should see that legs 2 and 3 are split by the suspected target zig during the ownship turn. We can guess that the target zig early in ownship leg 2 has mistakenly caused the algorithm to predict a target turn during the ownship turn. So, we will merge target legs 2 and 3:

2. Open the **Scenario (121200.00)** in the **Outline** view and you will see the bearing data listed, followed by our 4 target legs (**Tgt-x**).
3. **Ctrl-click** on legs **Tgt-2** and **Tgt-3** to select them.
4. Right-click on either one and select **Merge Straight Leg Forecasts**.

   As soon as you do this, the target legs will merge, the **Maintain Contributions** view will update, and the calculated solution will disappear from the plot because it's no longer valid.

5. Click on the **Calculate Solutions** button to rerun the solution. You will see that the new solution looks quite good.

Playing with legs

Once we're confident in the set of target legs we can move on to capture more of the analyst's knowledge and refine the target track.

Introduce speed constraint

Fortunately, the ship's log for the subject vessel gives a broad indication of the target's speed during the engagement. We will now add it to our scenario.

Add Speed constraint

There are a number of ways to add a speed constraint:

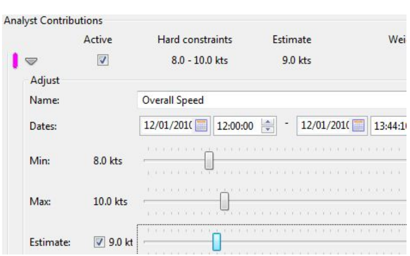
- At the per-leg level we can expand a **Straight Leg Forecast** and enter **min/max/estimate** values for course or speed for each leg.
- We can apply a speed constraint to the whole engagement.

To do the latter:

1. In the **Outline** view, right-click on the **Bearing Data** item and select **Add speed forecast for period covered by [Selected Legs]**. ☐
2. When the **New contribution** dialog box opens, enter **Overall speed** as the name of this contribution. ☐
3. Click on **Ok** and, since the addition of the new contribution means that the existing solution is no longer valid, you will see the target solution disappear. ☐

Specify speed constraint

The exercise observer recorded that the target was doing around 9 knots during the whole exercise, so we must enter this constraint to allow for speed keeping errors:



1. In the **Maintain Contributions** view, expand the **Speed Forecast** control. ☐
2. Enter a **minimum speed** of **8 knots**. ☐
3. A **maximum speed** of **10 knots**. ☐
4. An **(optional) estimate** of **9 knots**. ☐
5. Click on the **Calculate Solution** button. ☐

In the **Performance** tab you will see the effectiveness of both the **Bearing Data** and the **Overall speed** estimates. The bars are shaded according to the respective color-coded constraint.

Generate manual TMA solution

We will now use this information to develop a manual TMA solution.

1. In the **Outline** view, right-click on the **TMA solution** and select **Convert to Composite Track (legs)**. ❏

   A manual solution (**Track:121200.00_0**) will now appear in the **Outline** view, and the auto TMA solution will be hidden. ❏

Mark the new solution as a secondary track by:

2. Left-clicking to select it in the **Outline** view. ❏
3. Clicking on the **2** in the **Outline** view toolbar. ❏

We are now ready to manually adjust the solution.

Tune manual TMA solution

To manually tune the TMA solution:

1. In the **Outline** view, ensure the track is selected. ❏
2. Click on the **Drag Track Segment** button on the main application toolbar and the **Bearing Residuals** window will open. ❏

Note: if the **Bearing Residuals** window is empty, open the **Track Tote** view and check that your primary track is **Ownship** and your secondary track is **121200.00_0**. If they aren't, assign these accordingly in the **Outline** view.

Alternatively, if the ownship sensor cuts are not visible, you will need to clear the **Only draw dots for visible sensor points** button in the **Bearing Residuals** view toolbar.

1. Now, in the **Bearing Residuals** view, click on the **Shear** button. ❏
2. In the plot area, begin dragging track segments to minimise the bearing errors. ❏

Don't worry if your track segment dragging results in a mangled solution, you can delete the manual track from the **Outline** view and generate a new one from your existing solution.

Merge track segments

When you are finally happy with your solution it's time to merge the separate track segments into a formal track.

1. In the **Outline** view, expand your manual TMA solution.
2. Right-click on the **Track Segments (5 items)** and, from the popup list, select **Merge all track segments**.

You now have a track for the subject vessel that can be used for further analysis.

Mark your answers

To check how well you did:

1. In the **Outline** view, select **Track:Subject**.
2. Click on **Reveal Selected Items (make visible).**

This is the actual Truth Track; so, how does it compare against your track?

If you'd like a more quantitative score, produce an **XY Plot** of range between your solution and the truth track:

1. **Ctrl-click** on both tracks to select them.
2. Right-click and select **View XY Plot**.
3. In the **View time-variable plot** dialog, select **Range**.
4. Click on **Ok**.

You can now see a plot of how well you did.

Ideally, you should be able to achieve an accuracy of around 200 yds, which is not bad when we're using sensor data at around 20000 yds.

Conclusion

And that concludes Debrief's Semi-Automatic Track Construction and the end of this tutorial. Don’t forget that we have a comprehensive user manual to help you.

Good luck!

Signed: ________________ Date: ___________________