NavView User Guide – Node Dashboard Module

Document: 4DN_NVUG_M05_01B
Release: 01
Revision: B
Released: 4/25/2025
4D Nav, LLC

REL	REV	ISSUE DESCRIPTION	PREPARED	REVIEWED	APPROVED	DATE
01	B	Sections 4.4.1, 2, 3 and 4 added. Section 6.2 added	SW	AS	GAW	April 25, 2025
01	A	Initial release	SW	GAW	GAW	April 24, 2025

Table of Contents

1 INTRODUCTION

Node Dashboard is a Module for 4D Nav's NavView navigation software package which is designed to seamlessly integrate with subsea node deployment systems to aid in the positioning and logging of node locations.

Node Dashboard uses other components of NavView, such as vehicle positioning, and guidance calculations.

1.1 PREREQUISITES

The following is required to successfully operate Node Dashboard:

- SQL Server database connection using NavView Database Services

Note: Remote NavView stations require Database Services to be enabled and connected to the project SQL Server.

- A connection to the client node recorder system (if applicable)
- A position source and elevation source for each node deployment ROV

1.2 NETWORKED OPERATION

Node Dashboard takes control of NavView guidance calculations to guide ROVs towards their targets.

If operating in a networked environment with remote NavView stations, the position and guidance information is shared across the network for display. Refer to ***Network Services*** and ***Guidance Calculations*** sections in the NavView User Guide.

2 OVERVIEW

All configuration, views and utilities for Node Dashboard can be accessed from the dedicated Node Dashboard tab on the main NavView ribbon.

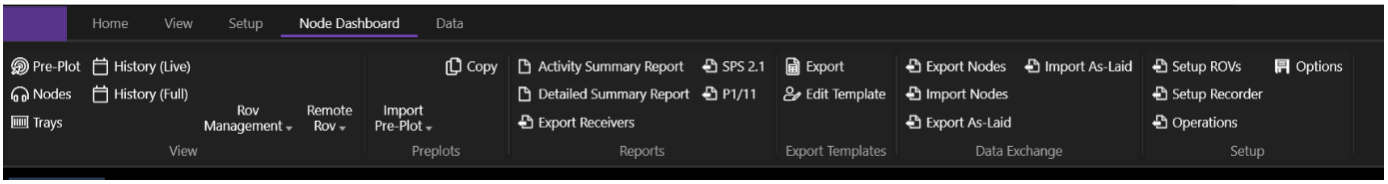


FIGURE 1 NODE DASHBOARD RIBBON

3 SETUP

This describes the Setup section in Node Dashboard.

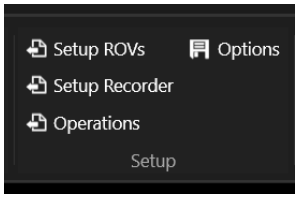


FIGURE 2 NODE DASHBOARD - SETUP

3.1 ROVS

Note: ROVs to be used for node deployment in Node Dashboard must first be added as a vehicle in NavView. Refer to ***Vehicles*** section in the NavView User Guide.

ROVs in Node Dashboard will:

- Create a Guidance Calculation with the name of the ROV
- Keep track of data sources to be logged and sent to the recorder

3.1.1 ADDING AN ROV

1. Click the **Setup ROVs** button on the **Node Dashboard** ribbon or select **ROVs** in the Node Dashboard section of Explorer to open the Configure ROVs page, see Figure 5.

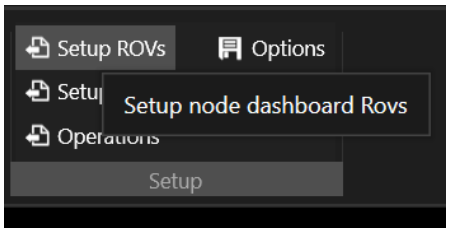


FIGURE 3 SETUP ROVS – NODE DASHBOARD RIBBON

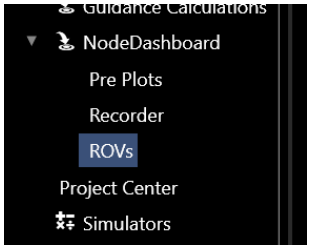


FIGURE 4 SETUP ROVS - EXPLORER

2. Click the  to add a node deployment ROV configuration this will open the configuration dialog.

3.1.1.1 ROV TAB

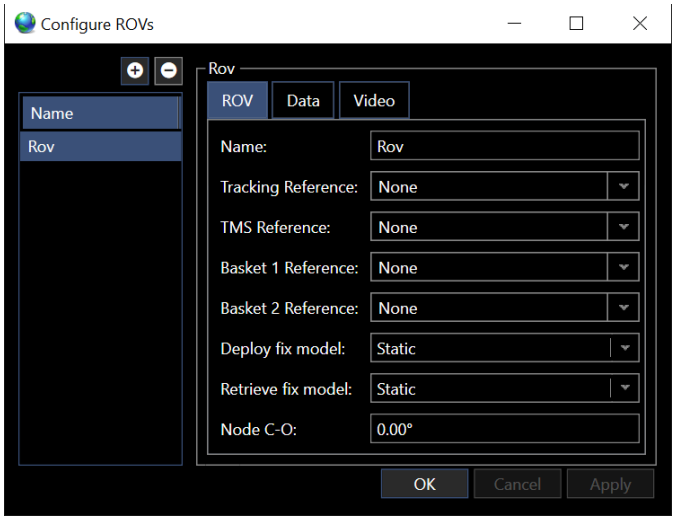


FIGURE 5 CONFIGURE ROVS – ROV TAB

- **Name:** The ROV will have the default name of "Rov", enter name to match the associated NavView vehicle.
- **Tracking Reference:** The source for the guidance calculation that will navigate the ROV pilots to the proposed node location. The reference should be an offset that represents the position of the node. The Tracking Reference to be used must be added to the NavView Vehicle as a Static Offset to be available for selection.
- **TMS Reference:** TMS position source from within NavView which will typically be set up as a Vehicle in NavView positioned by a beacon.

  Can be set up to automatically set the ROV's guidance calculation to navigate back to its TMS
- **Basket Reference:** Basket position source from within NavView which will typically be set up as a Vehicle in NavView positioned by a beacon

  Can be set up to automatically set the ROV's guidance calculation to navigate back to its Basket
- **Fix Model:** Can be set to either Static (default), Dynamic or Snapshot, see section 7.3 for more details on operation
  - **Dynamic** fix model enables fixes to be collected on the fly, while ROV is in motion while installing Nodes
  - **Static** fix is used if the ROV will be on seabed during the fix.
  - **Snapshot** represents a fix from a single instance in time – appropriate if an INS device is being used and the position output is already filtered
- **Node C-O:** Allows you to calculate the node heading relative to ROV heading

3.1.1.2 DATA TAB

Node Dashboard supports automatic logging of 10 different data sources when fixes are being recorded. Which item is selected as primary is important as this determines the node position sent to the recorder system.

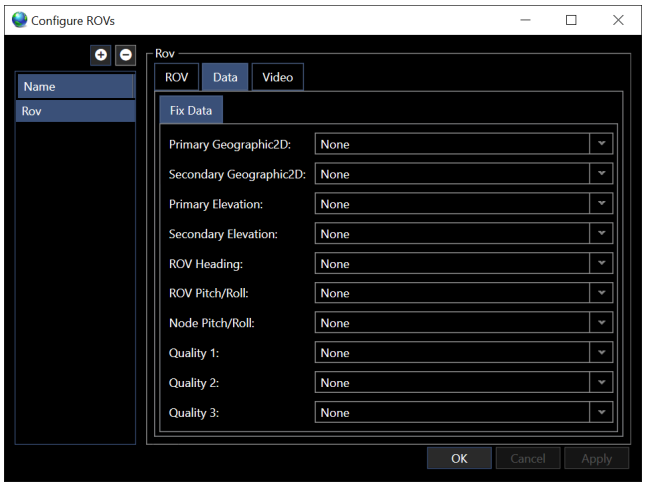


FIGURE 6 CONFIGURE ROVS - DATA TAB

- **Primary Geographic2D (Required)**: The position source representing the node location logged during the deployment that will be sent to the Recorder system.
- **Secondary Geographic2D:** The position logged for this data source during the deployment is averaged and used in reports to compare against the primary position for QC purposes
- **Primary Elevation (Required):** The elevation logged that will be sent to the recorder
- **Secondary Elevation: T**he elevation logged is averaged and used to compare against primary elevation for QC purposes
- **ROV Heading:** The heading logged and averaged and used in reporting
- **ROV Pitch/Roll:** The pitch/roll logged and averaged and used in reporting
- **Node Pitch/Roll:** The Node pitch roll is not averaged with the rest of the data. The surveyor will take an instantaneous grab of the data from this source once the Node has been set by the ROV. This allows for the average fixing to commence as soon as the ROV is in position, while the ROV is still positioning the node
- **Quality 1, 2, 3:** User defined quality fields are available which can be used to log any type of numerical data for reporting purposes

3.1.1.3 VIDEO TAB

Node Dashboard uses the video module of NavView to access camera feeds to:

- Use QR Codes to detect node IDs

- Record screen shots of node placements

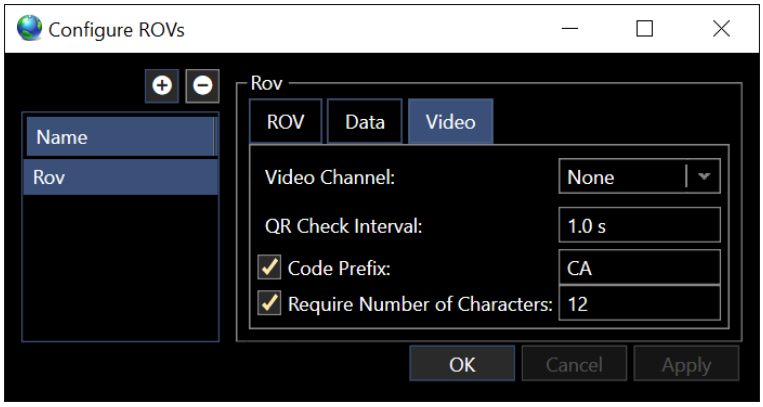


FIGURE 7 CONFIGURE ROVS – VIDEO TAB

- **Video Channel :** From drop-down list select the video channel to be associated with the selected ROV

Note: Refer to section 3.1.1.4 for setting up video channels.

- **QR Check Interval:** Enter interval to check for an updated QR code
- **Code Prefix:** The QR code is only accepted if it starts with entered characters and box checked
- **Require Number of Characters:** The QR code is only accepted if it contains the number of characters entered and box is checked

3.1.1.4 SETUP VIDEO CHANNELS

1. Click on the Video Setup button as shown in Figure 8 Video Setup – Setup Ribbon, to open the Video Channel Configuration window, see Figure 9. This window will allow the user to add, remove and configure video channels.

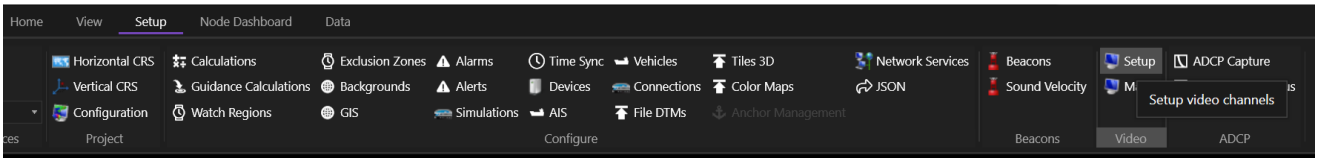


FIGURE 8 VIDEO SETUP – SETUP RIBBON

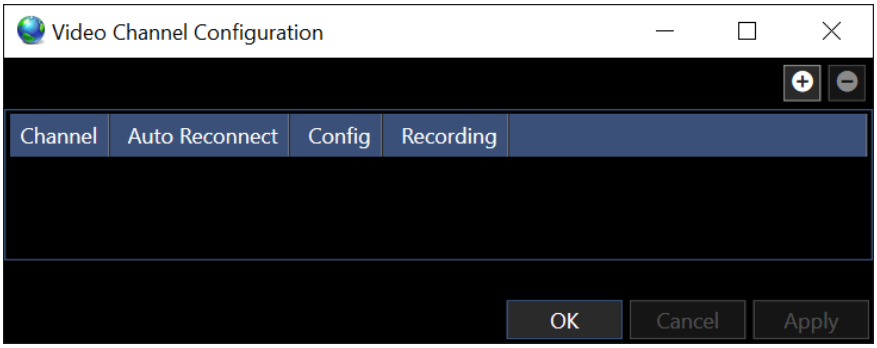


FIGURE 9 VIDEO CHANNEL CONFIGURATION

2. Add Video channel to the configuration by clicking the  button.

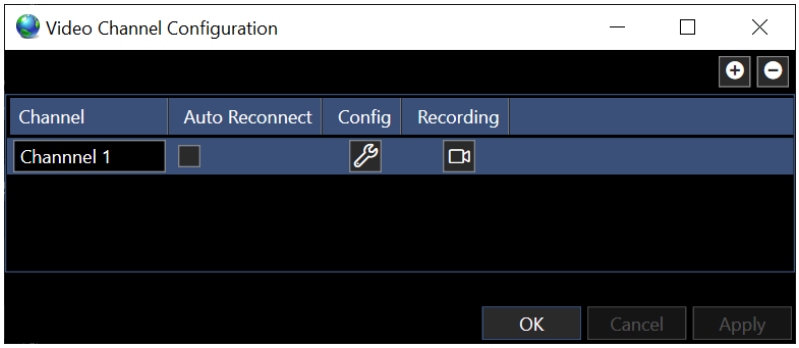


FIGURE 10 VIDEO CHANNEL ADDED

- **Channel:** Video channel should have a unique name that describes the source of the video
- **Auto Reconnect:** Checking box allows video to be automatically resumed when NavView starts
- **Config:** Video channel configuration button  for detailed config options, see Figure 11
- **Recording:** To enable/disable recording of short video clips

3.1.1.5 Configuring Video CHANNELS

Select the channel configuration button  to open the Configure Video dialog.

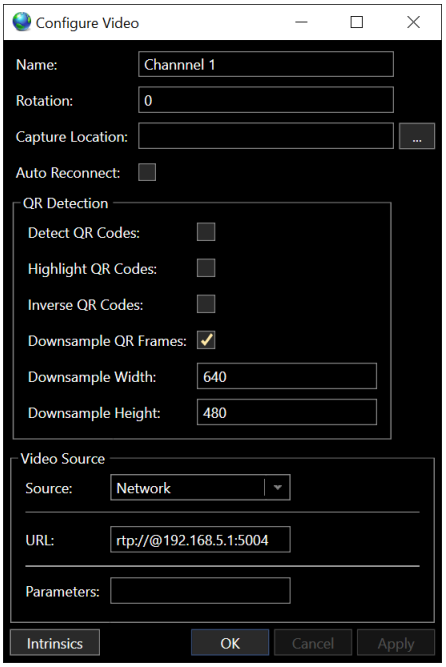


FIGURE 11 CONFIGURE VIDEO DIALOG

a. **Name :** Video channel should have a unique name that describes the source of the video
b. **Rotation:** Enter value to bring video in correct orientation
c. **Capture Location:** Click the browse button ... to navigate to the folder where the video clips/stills will be stored. The selected folder will be displayed in the box
d. **Auto Reconnect:** Checking box allows video to be automatically resumed when NavView starts
e. **QR Detection:** QR codes are 2D Barcodes which can be encoded with various kinds of string data such as URL, number identifiers, or plain text. NavView's video module can detect such codes. The QR code will be highlighted with a red square on the video window, along with its identifier. The QR identification results may be accessed by other NavView modules such as Node Dashboard.

FIGURE 12 QR CODE EXAMPLE

- **Detect QR Codes:** With the Detect QR Codes option enabled, the system will search for QR codes in each frame while the channel is active. When the option is disabled, all QR codes are ignored
- **Highlight QR Codes:** When enabled QR code will be highlighted with a red square on the video window
- **Inverse QR Codes:** Select this option if QR codes are printed as white on a black background instead of the standard black on white
- **Downsample QR Frames:** Enable to downsample frames for detection
- **Downsample Width:** To adjust downsample Width
- **Downsample Height:** To adjust downsample Height

f. **Video Source:** Select the source the video will be streaming from. The options include Device, File and Network

- **Device Source:** This option refers to a physical device connected to the computer.

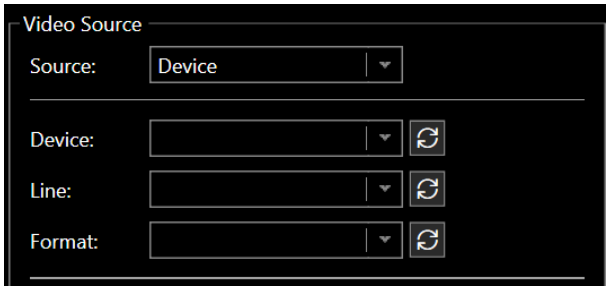


FIGURE 13 VIDEO SOURCE – DEVICE

- **Device:** Physically connected devices will be available to select from in the drop-down list. Click the  to refresh Device list
- **Line:** Some devices support multiple 'Lines'. This may include a PCI video capture card with 4x video inputs. The user would first select the capture card as the source, and then the line setting is used to choose which source from that device to use. If a device is chosen which does not contain multiple line options, the line box will remain blank and can be ignored. Click the  to refresh Line list
- **Format:** Once the device source and option line are selected the system will scan the device for the various options available to configure it. These options will then be available in the Format drop-down list. Formats are device dependent but would include such things as resolution, aspect ratio, and output frame rate. Click the  to refresh Format list

- **File Source:** This option is used to play video from a file for display in NavView. This could be used for quick review of video which has been captured. Click the browse button  to navigate to the video file to play

FIGURE 14 VIDEO SOURCE – FILE

- **Network Source:** User can specify the address of the network the video is on. NavView supports most common streaming technologies such as RTSP, RTP, or UDP streaming

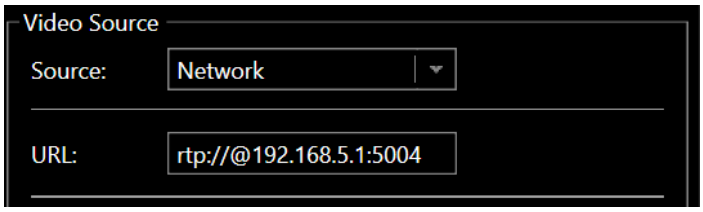


FIGURE 15 VIDEO SOURCE – NETWORK

Note: Refer to the device user manual for further details on interfacing it within NavView.

- **Parameters:** Parameters are advanced settings that correspond to those settings in the MFReader registry. Set something like file.buffer_max=3.0 in the parameters field
- **Intrinsics:** This is needed if you are using your camera feed for any type of measurements like photogrammetry. Not currently used by NavView

3.2 SETUP - RECORDER

Node Dashboard connects to the node recorder system using the Raw ASCII device. Once the Raw ASCII device has been added to NavView, it is selected from the Node Dashboard configuration pages, to be used with Node Dashboard.

Note: This section applies for system where a node recorder system is used. This is optional as some client deployments will not have a live feed from NavView.

1. Add a Raw ASCII device in NavView, refer to *Devices* in NavView User Guide.
2. Click the **Setup Recorder** button on the **Node Dashboard** ribbon or select **Recorder** in the Node Dashboard section of Explorer to open the Configure Recorder page.

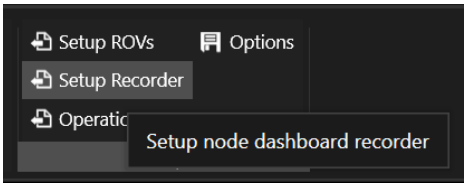


FIGURE 16 SETUP RECORDER – NODE DASHBOARD RIBBON

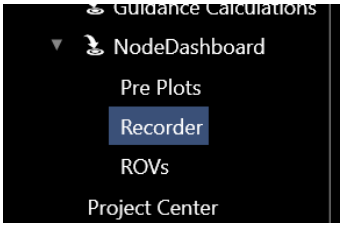


FIGURE 17 SETUP RECORDER – EXPLORER

3. Select the Recorder system type from the available options.

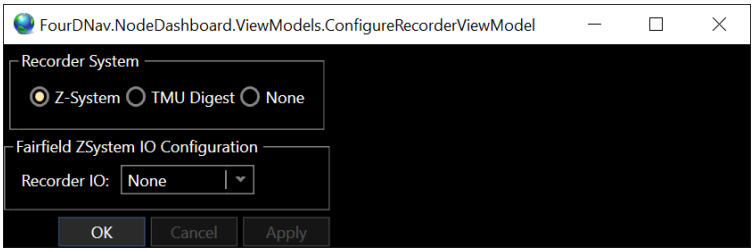


FIGURE 18 SETUP RECORDER – CONFIGURATION DIALOG

4. Select the Raw ASCII device that has been configured to connect with the recorder from the Recorder IO drop-down list.

3.3 SETUP - OPERATIONS

An Operation is a reference used to group a series of deployments or recoveries for reporting.

The Manage Operations dialog can be accessed from the Setup section of the Node Dashboard ribbon.

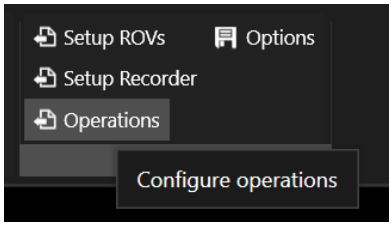


FIGURE 19 SETUP - OPERATIONS

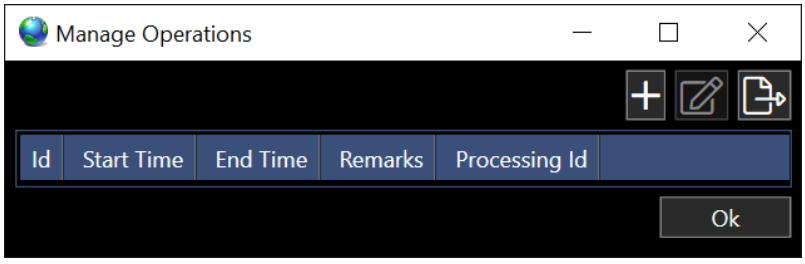


FIGURE 20 SETUP - MANAGE OPERATIONS DIALOG

3.3.1 ADD AN OPERATION

1. Click the button to add a new operation. This will bring up a dialog where remarks and processing ID can be entered.

Note: Remarks is automatically applied as the current local date and time.

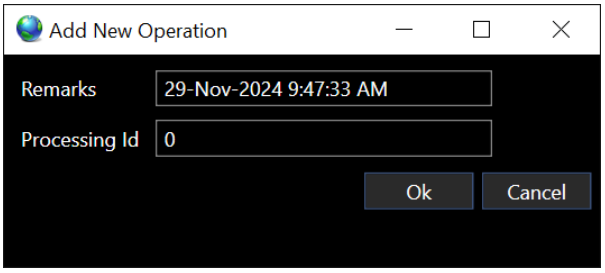


FIGURE 21 ADD NEW OPERATION DIALOG

2. Click Ok to add the operation or cancel to abort the process.

Note: An ID is automatically assigned to the Operation. An Operation cannot be deleted.

3.3.2 EDIT AN OPERATION

1. Select an Operation in the Manage Operations page.
2. Click the to open the Edit Operation dialog, see Figure 22.

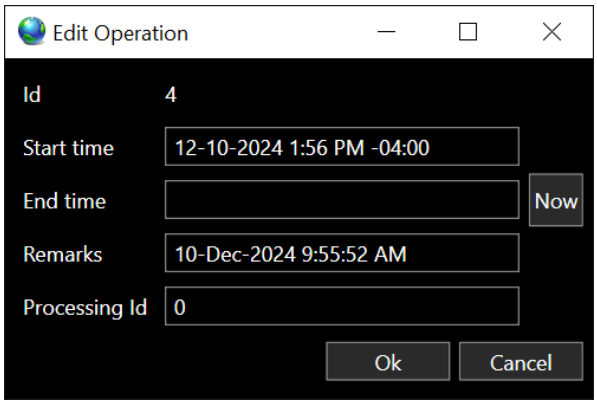


FIGURE 22 EDIT OPERATION DIALOG

Operations have the following properties:

- **Start Time:** UTC date and time when the operation starts
- **End Time:** End UTC date and time of this operation. To select the current time for the end time, click the **Now** button. To clear the end time so the operation is not set as complete, delete the time in the End time field

- **Remarks:** Allows the user to enter text to provide information about the operation. Remarks is automatically populated with the current local Start date and time
- **Processing ID:** A value associated with the operation for use when exporting to other systems

3.3.3 GENERATE A REPORT

Generate a .json formatted report which can be exported to Multiseis.

1. Click the  button to open the Operation Report Configuration dialog.

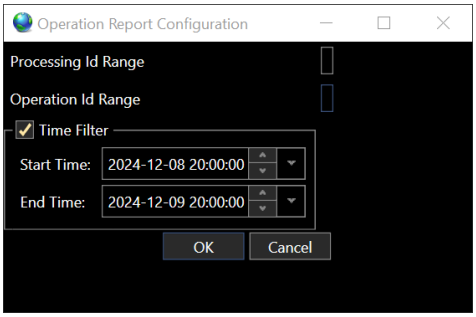


FIGURE 23 OPERATION REPORT CONFIGURATION DIALOG

2. The processing ID and operation ID range boxes can be used to filter the data to be used in the report. A single value can be entered, a comma separated list, or a range can be specified.
   - Enter 1 to export Id 1
   - Enter 1-10 to export all operations with ID 1 through 10
   - Enter 5,6 to export operations with Ids 5 and 6
   - Enter 1, 5-10 to export operation 1 and operations with Ids 5 through 10 (skips 2, 3 and 4)
3. A time filter can be setup to create a date and time window for export by selecting the Start time and End Time. This will export all operations within the set time window.
4. Click Okay to generate the report.

3.4 SETUP – OPTIONS

The Options setup provides additional configurations used in Node Dashboard.

Select Options in Setup of the Node Dashboard ribbon. This will open the Settings dialog, see Figure 25.

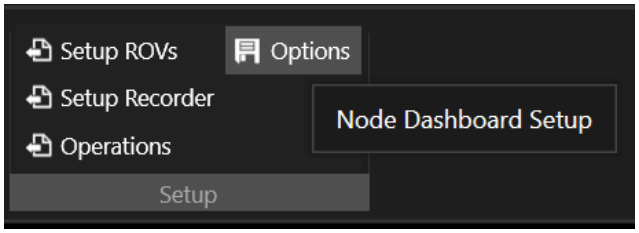


FIGURE 24 SETUP – OPTIONS

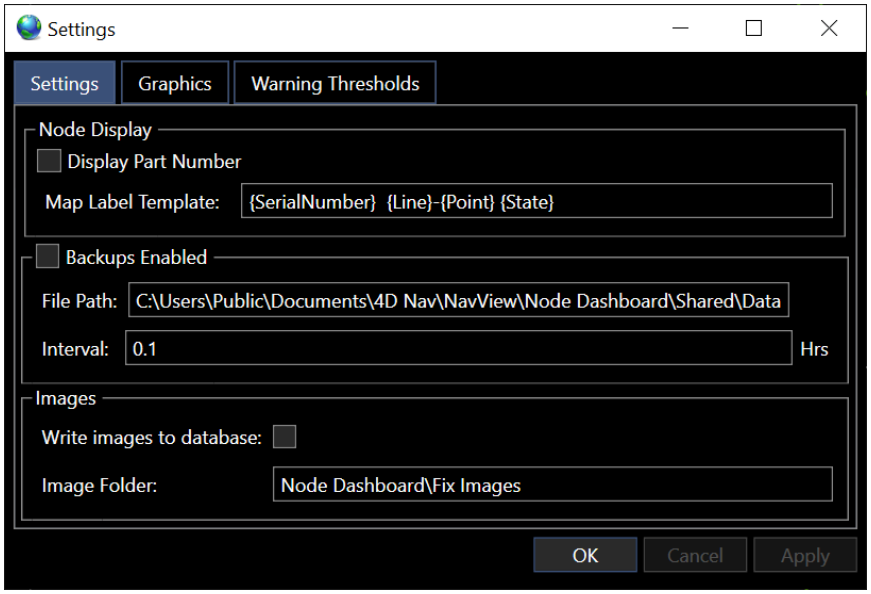


FIGURE 25 OPTIONS – SETTINGS DIALOG

3.4.1 SETTINGS TAB

- **Node Display:** When enabled, nodes will be listed as [Serial Number] [Part Number]. When disabled, nodes will be displayed as Serial Number only
- **Backups Enabled:** When enabled, the Node Dashboard database will be backed up to the specified File Path. The backup Interval is in hours

Note: In order to use the automatic backup feature, the NavView system must have Microsoft SQL Server command line utilities installed and available on the path. This is installed automatically with an install of SQLExpress, however, if the server is running on a different PC, the install may need to be done separately. Additionally, enhanced backup script must be installed on the database sever via SQL scripts. Please contact 4D Nav support to get assistance with setting up the scripts and ensuring you have the SQL command line utilities installed.

- **Images:** When enabled, Node fix images will be sent to the specified Image Folder

3.4.2 GRAPHICS TAB

In the Graphics tab, the user can define how the Pre-plots, Nodes and As-Laid are presented on the Map view.

3.4.2.1 PRE-PLOT TAB

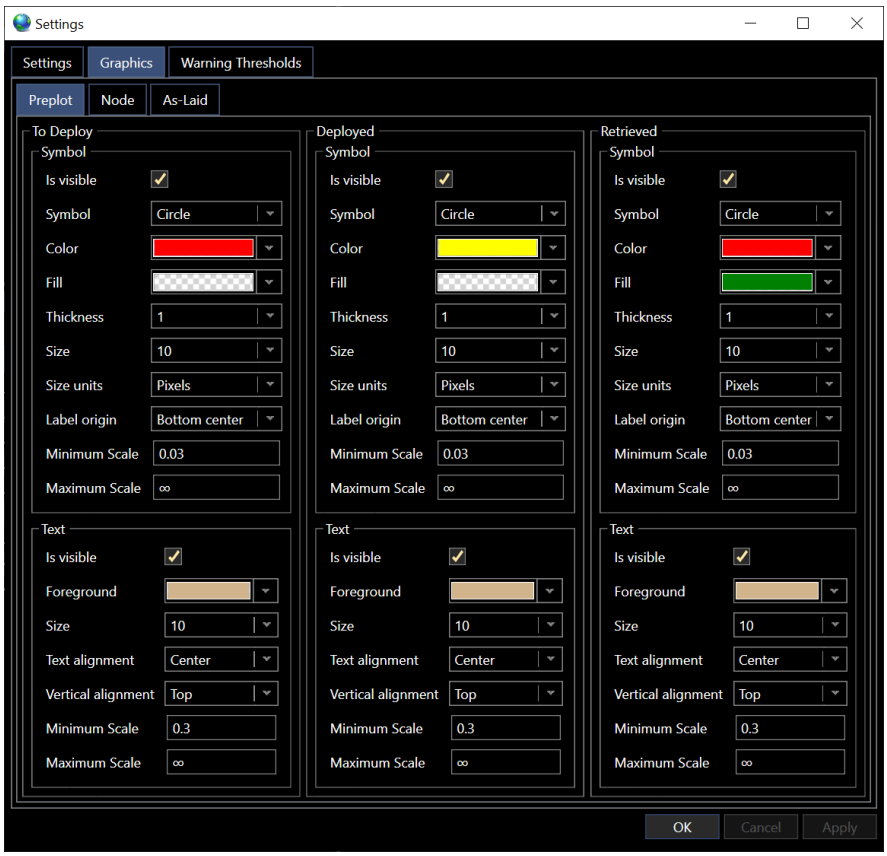


FIGURE 26 OPTIONS – GRAPHICS – PRE-PLOT

Size Units : The size unit setting is used in conjunction with the Size property to display the shape on the map. If the Size unit is set to Pixels, the shape will always be the same size on the map, no matter the zoom level, however with the Map Units option, the icon will show up scaled to the selected horizontal projection unit. This option allows for quick setup of range rings for navigation of the ROV within the defined tolerance.

3.4.2.2 NODE TAB

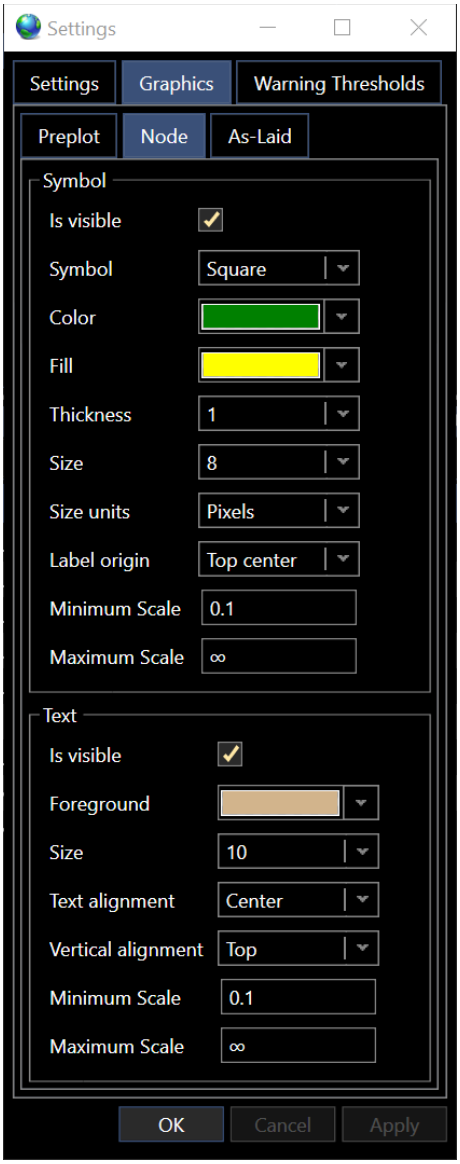


FIGURE 27 OPTIONS – GRAPHICS – NODE

3.4.2.3 AS-LAID TAB

FIGURE 28 OPTIONS – GRAPHICS – AS-LAID

3.4.3 WARNING THRESHOLDS TAB

Dialog to specify warning criteria for Node positioning during operations.

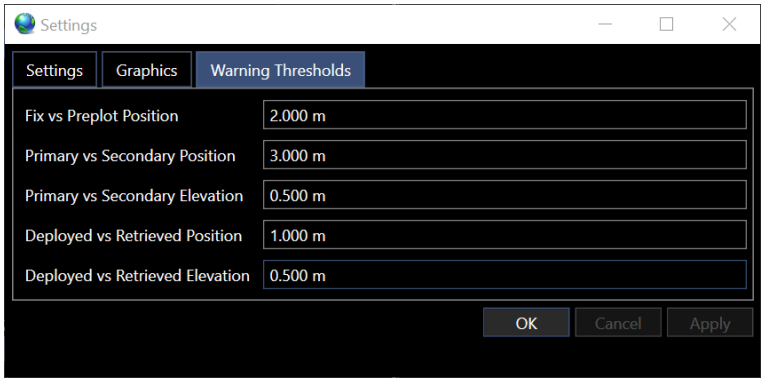


FIGURE 29 OPTIONS – WARNING THRESHOLDS

The warning thresholds are used to colour code the results in the As-Laid history window as well as the preplot details/as-laid details windows. Any results that are out of the specified tolerance will be highlighted in red.

4 PRE-PLOTS

Pre-plot positions refer to the target positions for node installations and used for navigation targets and for logging purposes.

The Pre-plot positions are imported into Node Dashboard and stored in a SQL Server database.

These positions come in a variety of standard file formats and custom csv files. Node Dashboard supports the following file formats for Pre-plot import:

- SPS 2.1
- SPS 0.1
- P1/11
- CSV: Line, Pt Number, Easting, Northing
- CSV: Line, Pt Number, Pt Index, Pt Code, Easting, Northing

Note: Database Services must be enabled prior to importing pre-plot file.

4.1 IMPORT PRE-PLOT POSITIONS

1. In the Pre-plots section of the Node Dashboard ribbon, click on the Import Pre-plot drop-down button

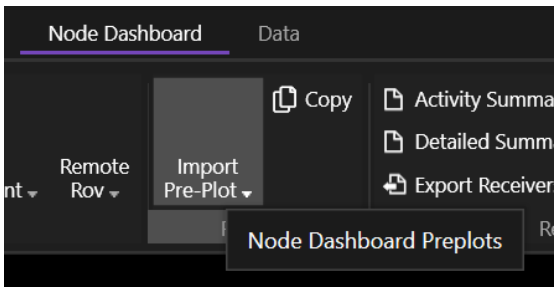


FIGURE 30 IMPORT PRE-PLOT

2. Select the appropriate file format.

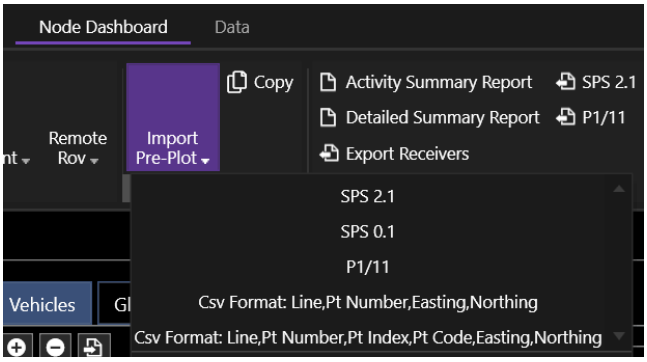


FIGURE 31 PRE-PLOT – FILE FORMATS

3. Select the file to import.

If the format of the file is correct, the Pre-plots will be imported and added to the database. A dialog will appear indicating the number of Pre-plots that have been imported.

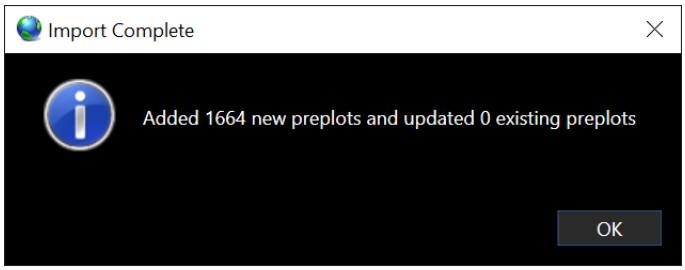


FIGURE 32 PRE-PLOTS IMPORT COMPLETE

4. Click Okay to accept.

4.2 UPDATING EXISTING PRE-PLOT DATABASE

If new pre-plot positions (file) are provided after a database has already been imported.

a. Follow steps for Importing Pre-plot Positions using the file with the updated design positions.
b. Node Dashboard will,
    - Read each record in the file being imported
    - Check the pre-plot points in the existing pre-plot table (i.e. database) for a match of the following;
        - Line Name
        - Point Number
        - Index
        - Code
c. If a point is found where all 4 items match, the design position for that point is updated in the database from the file being imported

Note: Only updates the target locations for existing points that have not already been deployed.

d. If a point is found where any one of the 4 items does not match, e.g. the Index a new point is added to the pre-plot database without changing any existing points

4.3 COPY PRE-PLOTS

The copy option lets the user copy a series of Pre-plots from one line and index. This will create a new copy of each Pre-plot at the next index on the same line and point for each one.

In the Pre-plots section of the Node Dashboard ribbon, click on Copy. This will open the Copy Pre-plots dialog, see Figure 34.

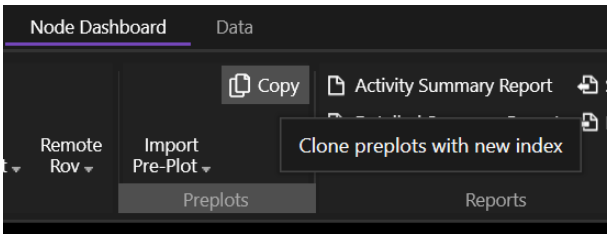


FIGURE 33 COPY PRE-PLOTS

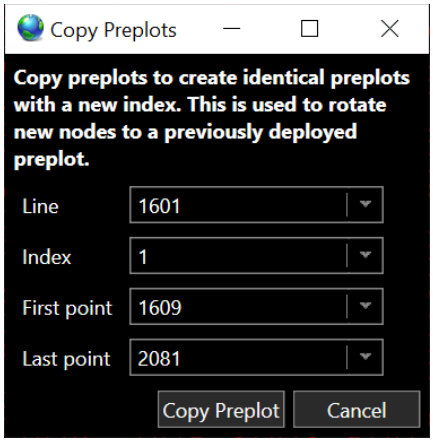


FIGURE 34 COPY PRE-PLOTS DIALOG

4.4 VIEWING PRE-PLOT DATA

To view the pre-plot data, select the Pre-Plot button from the View tab in the Node Dashboard ribbon.

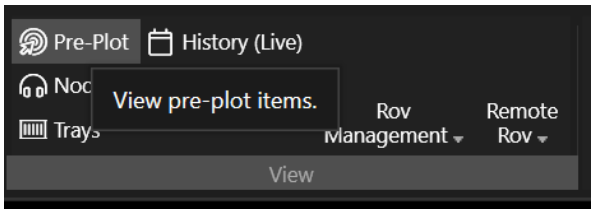


FIGURE 35 VIEW PRE-PLOT DATA – NODE DASHBOARD RIBBON

OR

Select Pre Plots in NodeDashboard section of the Explorer tree.

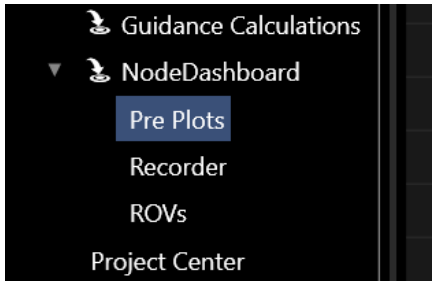


FIGURE 36 VIEW PRE-PLOT DATA – EXPLORER TREE

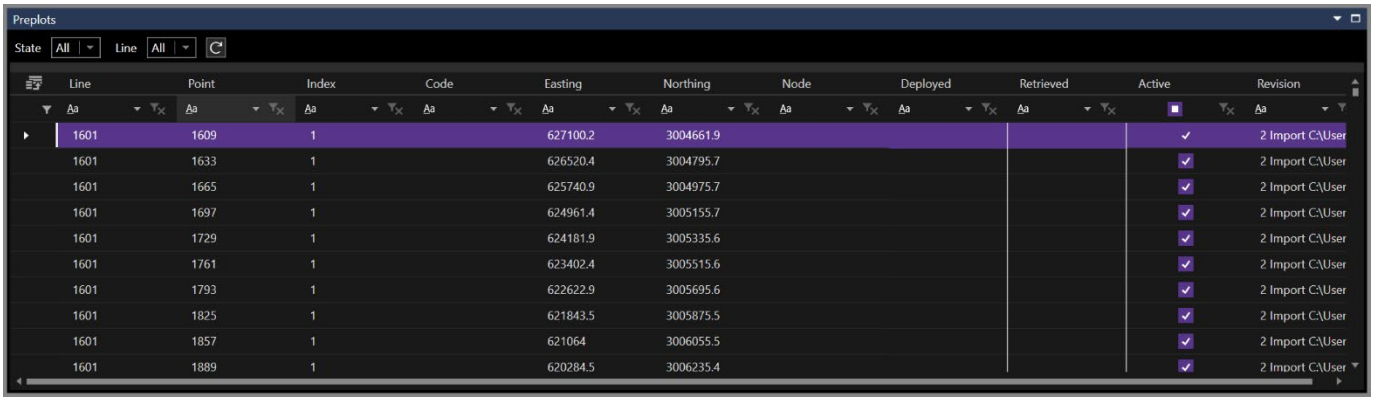


FIGURE 37 PRE-PLOTS DATA WINDOW

The Pre-plots window can be filtered by State and by Line number using the dropdown boxes at the top of the view. Additionally, there is a full featured data filtering interface in this data

table which allows filtering by any property including multiple simultaneously, in an excel-like interface.

Right clicking on a row in the Pre-plots window will open a popup menu showing available actions.

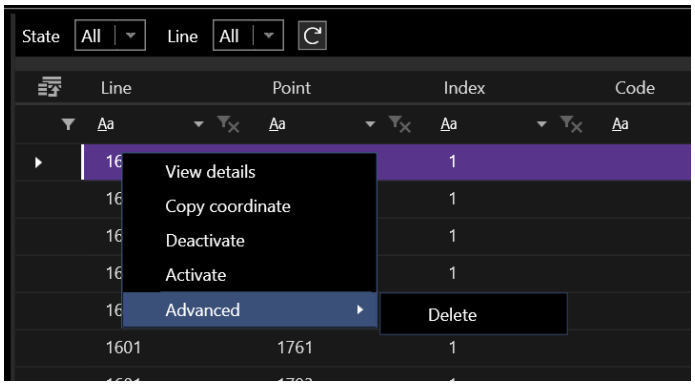


FIGURE 38 PRE-PLOTS WINDOW – POPUP MENU

4.4.1 VIEW DETAILS

The view details option will open up the Preplot Details dialog box for the selected preplot. This is done only for one preplot, the one under the cursor when the menu was opened.

4.4.2 COPY CORDINATE

Copies the coordinate of the preplot to the clipboard so that it can be pasted in other parts of NavView such as to create waypoints.

4.4.3 ACTIVATE/DEACTIVATE

Deactivating Preplots is a way to remove them from the actively used interface without removing from the project. Deactivated preplots are not displayed on the Map View or included in any exported reports, however they are still shown in the Preplots window so that they can be selected for reactivation.

4.4.4 DELETE PREPLOT

Use this option to delete a preplot from the project. Use with caution.

4.5 VIEWING PRE-PLOTS ON MAP

Once the Pre-plots have been imported they can be seen on the Map view.

1. Open the Map view by clicking on the **Map** button on the **View** tab ribbon.
2. Right click anywhere on the map and use the Move-To menu to navigate to the Pre-plots listing.
3. Select a line to move to the first Pre-plots on that line.

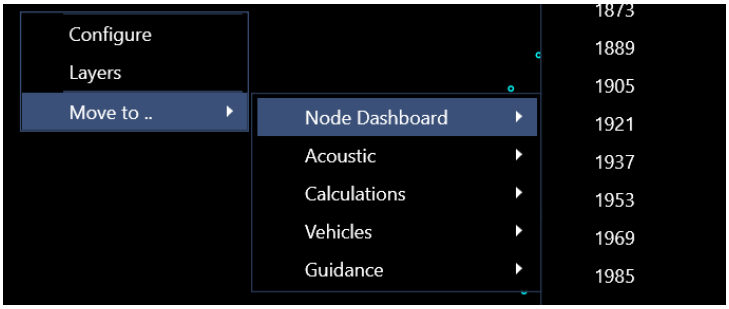


FIGURE 39 MOVE TO PRE-PLOT LINE

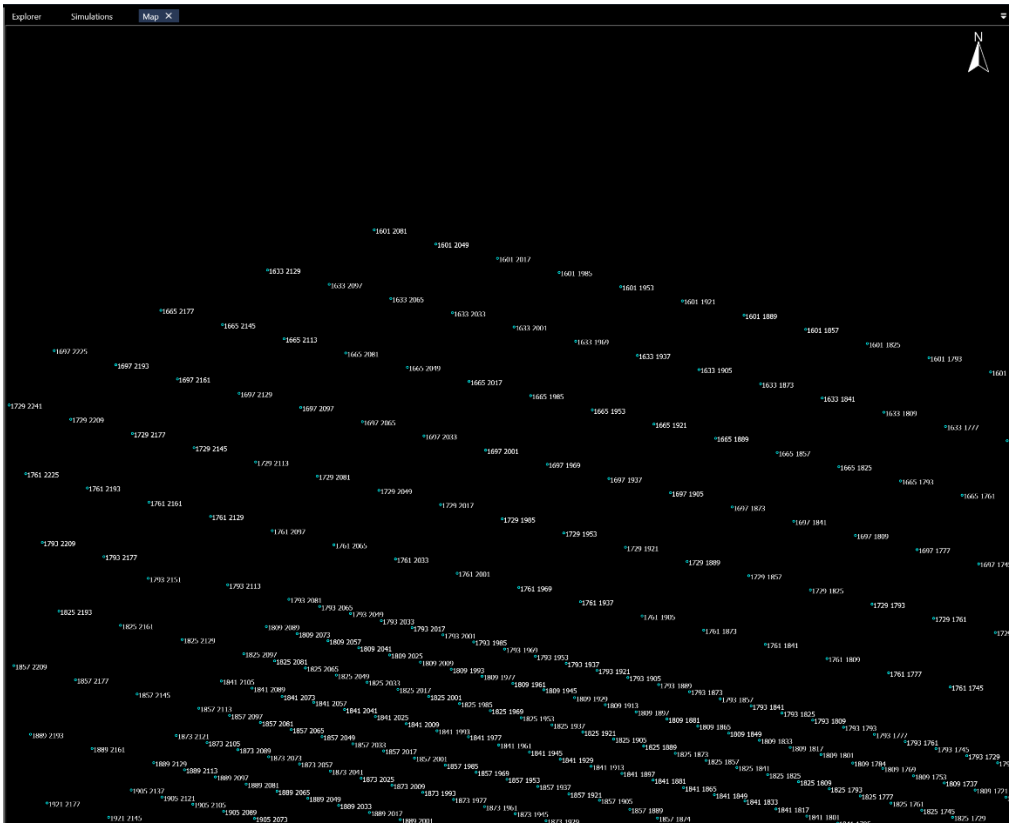


FIGURE 40 PRE-PLOTS MAP VIEW

Note: See Setup Options section 3.4.2 to configure how the Preplots are displayed on the Map view. There is a minimum/maximum map scale at which the Preplots and the labels will appear. If the Preplots do not appear, check the current map scale.

5 NODE MANAGEMENT

Nodes are added to NodeDashboard either by message from a recorder system, QR code detection, import (Data Exchange) or by manual entry.

Node Dashboard keeps track of the state of each node. The possible states of nodes are:

- Unknown
- Deployable
- Deployed
- Retrievable
- Retrieved
- Wet Stored

When nodes are started by a recorder system, a started message is sent to NodeDashboard with a list of nodes which are then set to the deployable state.

When a node is deployed, NodeDashboard sets it to the Deployed state in which case it cannot be deployed again, until it is retrieved.

Retrieved nodes can be redeployed at another location, they will show up in the Retrieved list but will be available for use when the ROV is in Deploy mode.

If new nodes are added by QR code detection or by manual entry they will also be set to deployable. Imported nodes can have a node state specified in the import file.

Note: Only the nodes whose state is Deployable can be removed from the Node list if there is mistake and needs to be edited.

5.1 VIEWING NODES

To view the list of available nodes.

1. Select the Nodes button in the View section of the Node Dashboard ribbon.

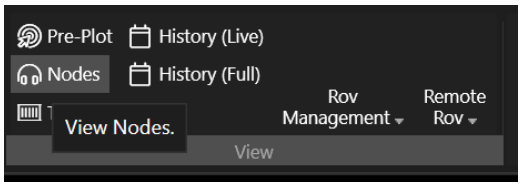


FIGURE 41 VIEW NODES – NODE DASHBOARD RIBBON

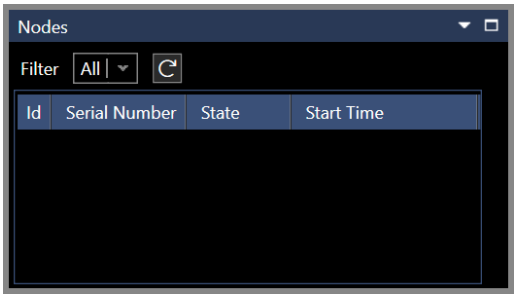


FIGURE 42 NODES VIEW WINDOW

2. The Node list can be filtered by selecting a node state from the drop-down menu.

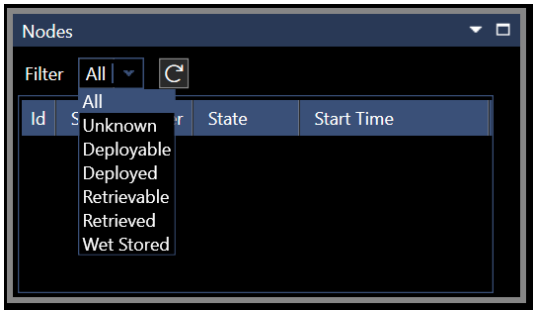


FIGURE 43 NODES VIEW FILTER OPTIONS

3. Click the  to refresh the node list from the database.

5.2 REMOVING NODES

From the Nodes window, existing nodes can be removed from the system.

This can be used if nodes were added for testing purposes or falsely detected QR codes.

To remove a node right click on the node in the Nodes window and select Remove Node in the pop-up menu.

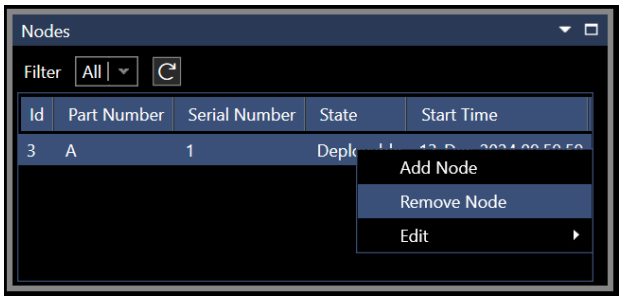


FIGURE 44 REMOVE NODE – NODES WINDOW

Note: Only Deployable nodes can be removed from the system.

5.3 EDITING NODES

The Serial number and/or part number and State of an existing node can be edited.

To edit a node right click on the node in the Nodes window and select Serial Number or State in the pop-up menu.

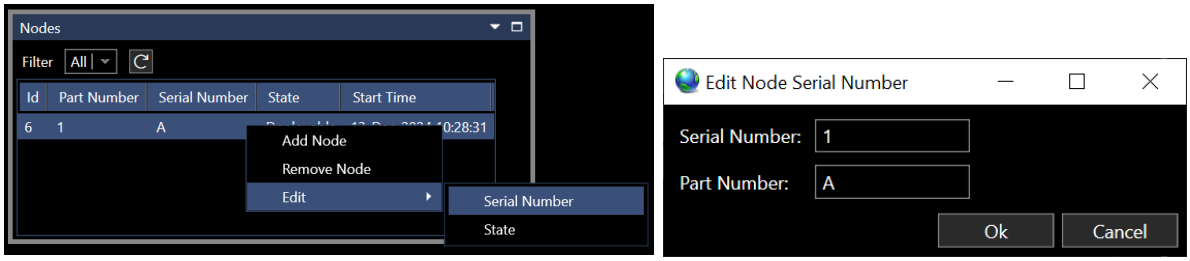


FIGURE 45 EDIT NODE SERIAL NUMBER

FIGURE 46 EDIT NODE STATE

5.4 MANUALLY ADDING NODES

New nodes can be manually added to the system by right clicking in the Nodes window list area and selecting the Add Node option.

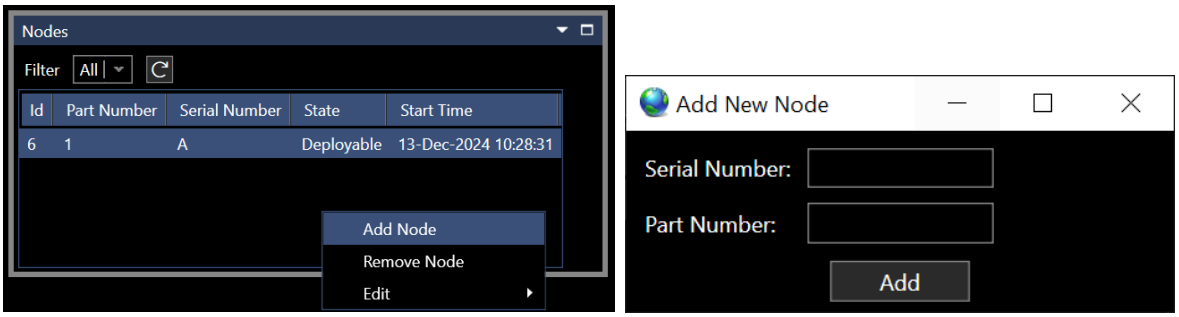


FIGURE 47 ADD NODE MANUALLY

In the Add New Node dialog specify a serial number and optional part number.

6 TRAY MANAGEMENT

The tray management window is accessed by clicking the Trays button in the View section of the Node Dashboard ribbon.

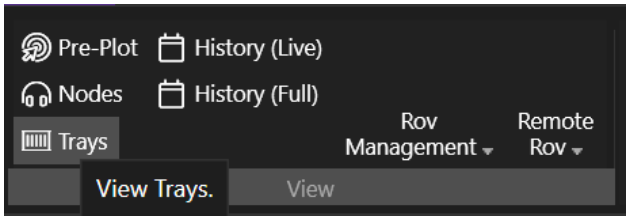


FIGURE 48 VIEW TRAYS – NODE DASHBOARD RIBBON

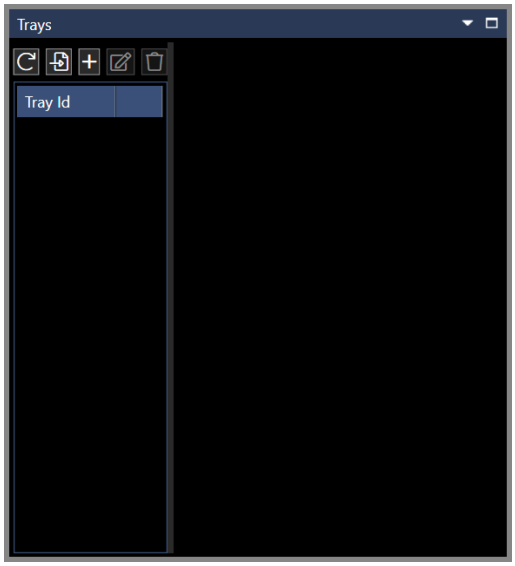


FIGURE 49 TRAY MANAGEMENT WINDOW

Tray references may be sent directly from an external system or created manually.

- **Request Tray Update:** If connected to an external system for tray management, this will request an update from that system and update the local list of trays and their states
- **Import New Tray:** Imports tray from a text file
- **Add New Tray:** Opens the Add New Tray dialog, creates a new tray and opens the Edit Tray window for the newly created tray

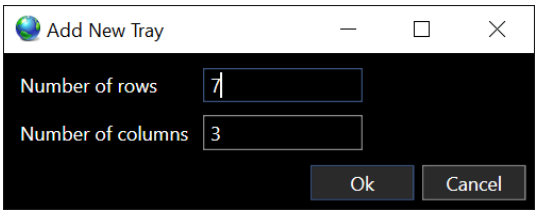


FIGURE 50 ADD NEW TRAY DIALOG

- **Edit Tray:** Opens the Edit Tray dialog for the selected tray

- **Remove Tray:** Removes the selected tray from the local system. Notifies a remote tray management system, if available, that the tray is empty

6.1 EDIT TRAY

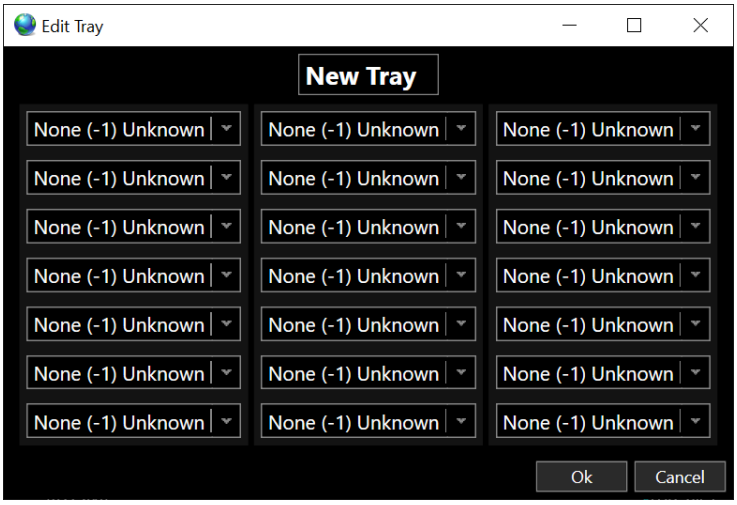


FIGURE 51 EDIT TRAY DIALOG

1. Tray name can be edited.
2. Add Nodes to the slots in the tray.

Note: If the tray name is changed, this essentially creates a new tray and moves all nodes to it which will break the link if any of the ROVs have that tray selected. In this case the tray will need to be reselected on each ROV.

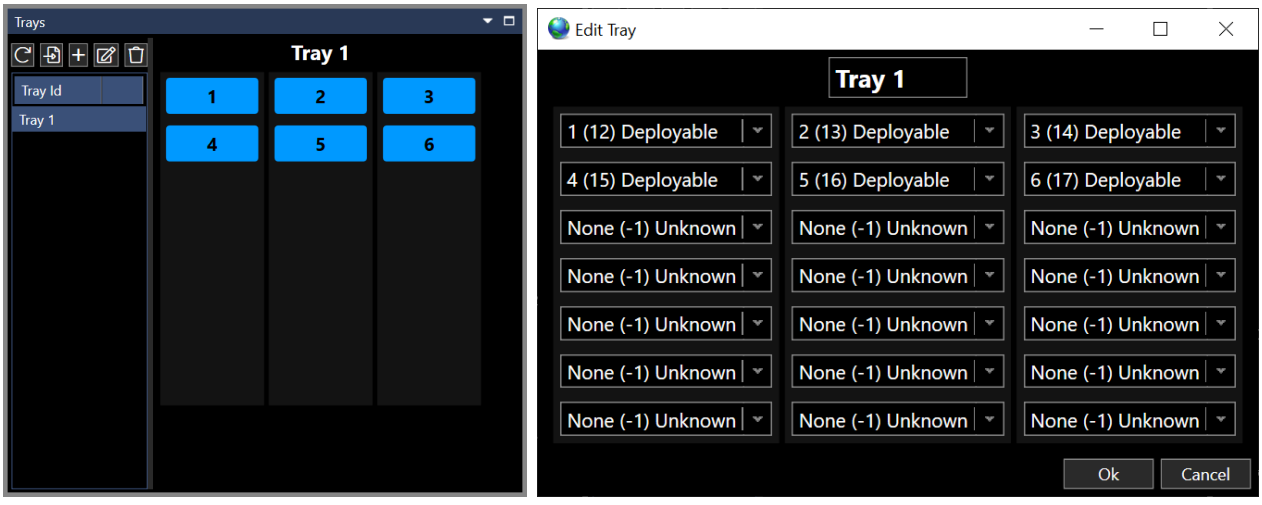


FIGURE 52 NODES ADDED TO TRAY EXAMPLE

6.2 IMPORTING TRAYS

Trays may also be imported into the system via a CSV file by clicking the Tray Import button  .

The Configuration of the import is controlled by the file trayimportSchema.json which is located within the local project configuration folder. The Tray Import schema file contents are shown as:

{"Separator":",","HasHeader":true,"NumFieldsPerNode":3,"SerialNumberIndexInNodeFields":1}

- **Separator:** By default, a comma is used, but another separator may be specified
- **Has Header:** If true, the first line in the import file will be ignored
- **NumFieldsPerNode:** Defaults to 3, this indicates how many columns in each row is occupied by a single node. This allows files to contain other node information but for Node Dashboard to select the serial number from within those fields for import

An example schema to bring in a file with just serial numbers as a comma separated file with no header is shown here:

{"Separator":",","HasHeader":false,"NumFieldsPerNode":1,"SerialNumberIndexInNodeFields":0}

7 ONLINE OPERATIONS

To deploy/retrieve a node, the ROV must be directed towards that location. The ROV navigation within NodeDashboard is controlled either from the ROV Management Views or from the map view.

7.1 ROV MANAGEMENT

Provides monitoring of the ROVs configured for the node deployment and controlling the node deployment and retrieval process.

1. Click the ROV Management button in the View section of the Node Dashboard ribbon.

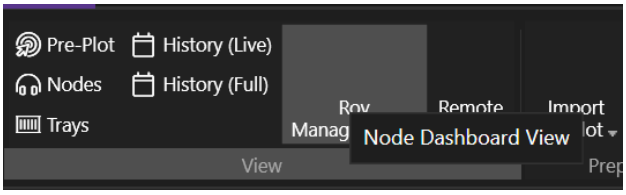


FIGURE 53 ROV MANAGEMENT – NODE DASHBOARD RIBBON

2. The drop-down button provides a list of available ROV's in the project. Click on the ROV to view in the ROV Management Window, see Figure 55.

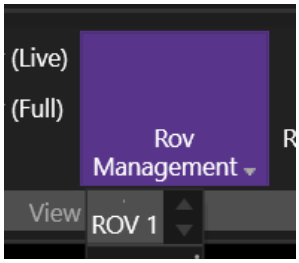


FIGURE 54 ROV MANAGEMENT – ROV SELECTION

7.1.1 MANAGE TAB

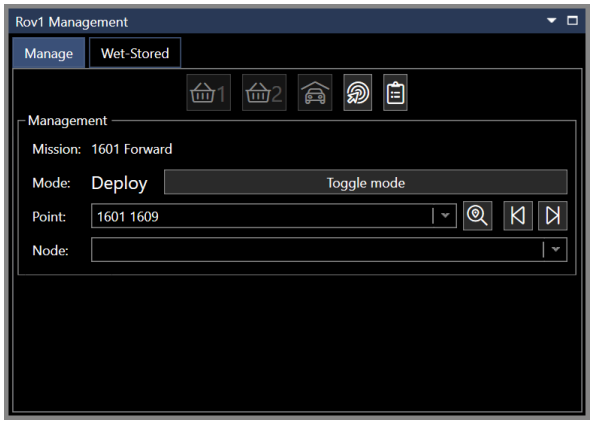


FIGURE 55 ROV MANAGEMENT WINDOW – MANAGE TAB

- **Navigate to Basket:** This tool will direct the ROV to the assigned basket as selected in Setup ROV's
- **Navigate to TMS:** This tool will direct the ROV to its TMS position as selected in Setup ROV's
- **Navigate To:** This tool will direct the ROV to a selected Pre-Plot position. Select the Pre-Plot position in the drop-down menu
- **Update Operating Parameters:** Opens Operating Parameters dialog to edit the parameters such as tray to use, lines to run etc.

7.1.1.1 OPERATING PARAMETERS

The operating parameters window allows the user to select an Operation currently in use, specify one or more lines and how they are to be deployed, and select a tray.

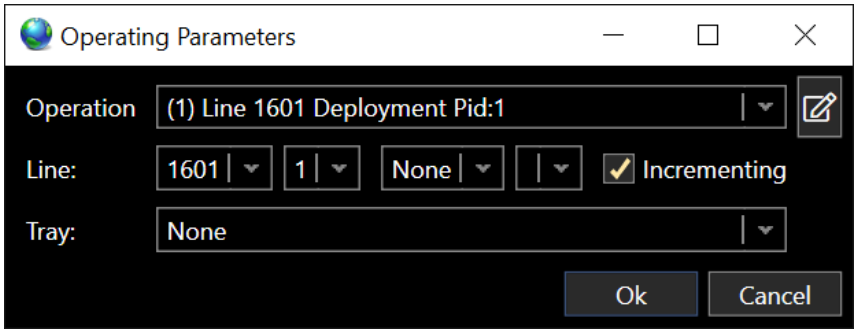


FIGURE 56 OPERATING PARAMETERS DIALOG

1. Click the Operating Parameters button  . Select an available operation from the drop down box. The default option is to have no operation selected. Only operations which are currently open, and do not have an end time are available for selection.

   Use the  to quickly access the operation management view. See section 3.3 for details on Operations.

2. Line selection.

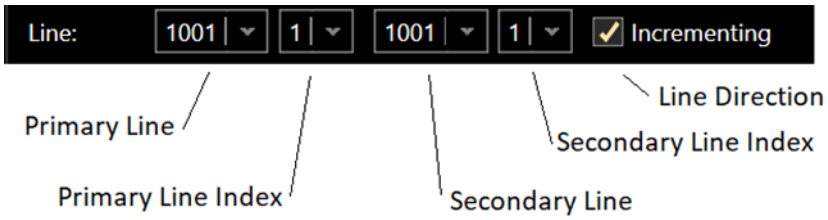


FIGURE 57 LINE SELECTION

If only one line is to be used, the Secondary line may be set to None.

For 2 line operation, the system will zig-zag between the two lines.

3. Select from available trays to assign to this ROV. A preview of the selected tray is shown at the bottom of the window. See section 6 for details on tray management.

7.1.1.2 MANAGEMENT PANEL

FIGURE 58 ROV MANAGEMENT PANEL

- **Mission:** Pre-Plot line and direction selected in Operating Parameters
- **Operation:** Operation selected in Operating Parameters
- **Mode:** Mode is either Deploy or Retrieve this can be changed using the *Toggle Mode* button
- **Point:** Select a pre-plot from the drop down list. The pre-plots in the list are limited to the lines assigned to the ROV. Also, a pre-plot can be selected using tools below
  - **Advance to closest logical pre-plot:** Selects the closest pre-plot from the list to the current position of the ROV
  - **Advance to Previous pre-plot:** Selects the previous pre-plot in the list
  - **Advance to Next pre-plot:** Selects the next pre-plot in the list
- **Node:** Select an available node from the drop down list. All deployable nodes are available in the drop down box. If a tray is assigned to the ROV a node can be selected by clicking on a node from the tray area, see Figure 59.

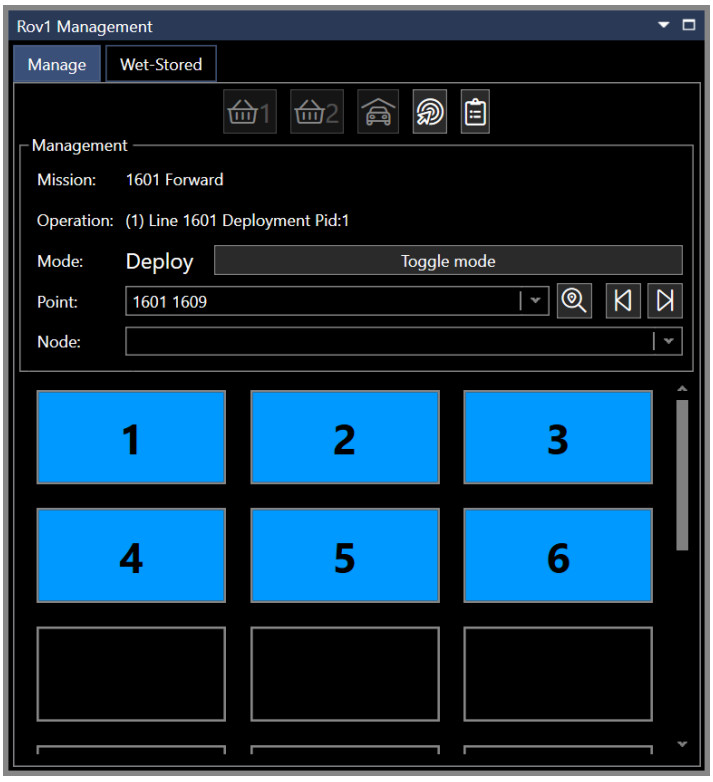


FIGURE 59 ROV MANAGEMENT WITH ASSIGNED TRAY

7.2 NODE ACTIVITIES

7.2.1 DEPLOY NODE

When the ROV is in Deploy mode and a node is selected, the lower portion of the management window will display the deployment menu.

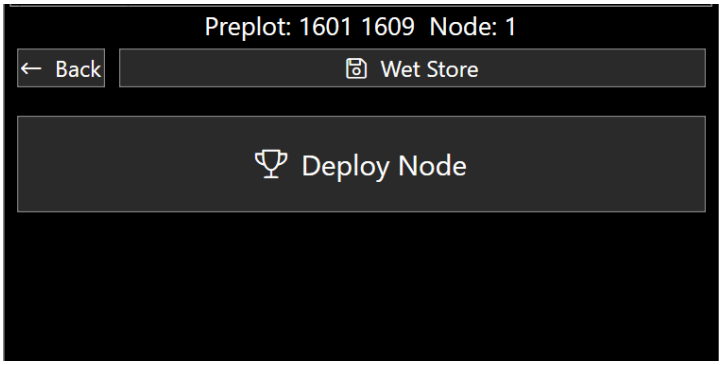


FIGURE 60 DEPLOY MODE MENU

The Deploy mode menu displays the node and pre-plot selected and allows the user to either wet store the node or deploy it.

Click  button to deploy the node. This will start the deployment process using the fix model selected for that ROV, see 3.1.1.1 and 7.3.

Once confirmed the deployed node will change to green in the tray display.

FIGURE 61 ROV MANAGEMENT – NODE 1 DEPLOYED FROM TRAY

The selected point will then automatically switch to the next logic pre-plot in the list so that the ROV can be directed to that location.

Clicking on the Toggle Mode button or the deployed node in the tray will cause the ROV to enter Retrieve Node mode and present the options for Retrieval, Move or Undo of deployed.

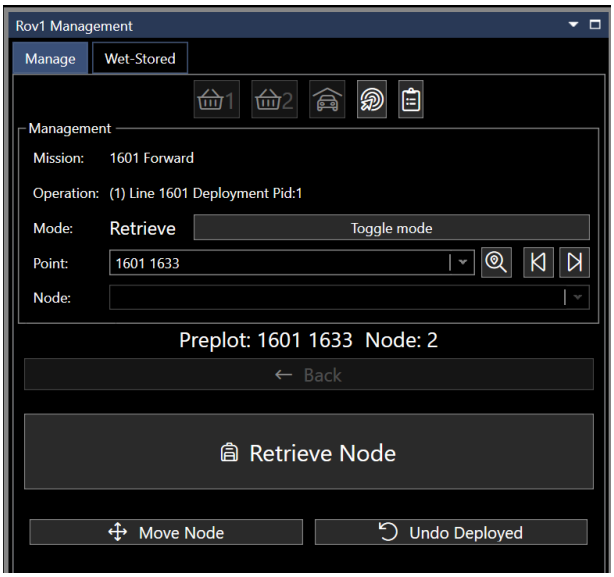


FIGURE 62 RETRIEVE MODE

A node can also be deployed from the Map view by right clicking on the pre-plot and selecting Deploy from the context menu. Select the ROV and node to deploy from the menu. This will activate the Deploy Node option in the ROV Management window.

FIGURE 63 DEPLOY NODE – MAP VIEW

7.2.2 UNDO DEPLOYED NODE

To undo a deployment, click the 

button in Retrieve mode, see Figure 62. This will remove the fix of the deployment and put the node back to the deployable state so that it can be used somewhere else.

Note: The ROV will still be in Retrieve mode so the screen will remain the same. Click the Toggle Mode button to redeploy the node on a different pre-plot.

7.2.3 MOVE NODE

If a node has been deployed but the fix was in the wrong location a new fix can be taken using the Move Node button in Retrieve mode, see Figure 62. This will take a new fix using the selected Deploy fix model and replace the existing fix.

7.2.4 RETRIEVE NODE

Clicking the Retrieve Node button, see Figure 62, will launch a fix using the selected retrieve fix model. Once a node is retrieved it will appear red in the tray.

A node can also be Retrieved from the Map view by right clicking on the deployed node and selecting Retrieve and ROV from the context menu. This will activate the Retrieve Node option in the ROV Management window.

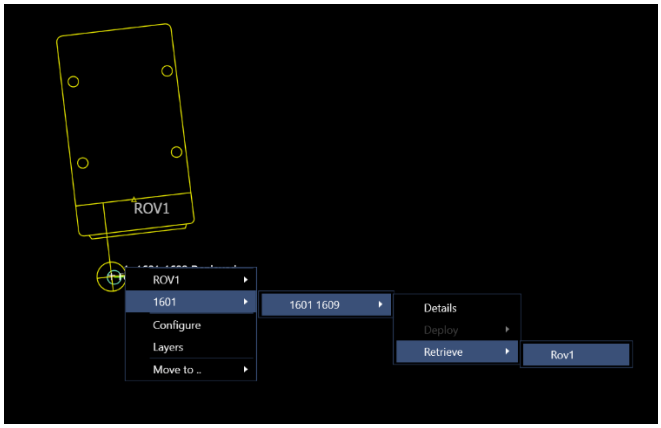


FIGURE 64 RETRIEVE NODE – MAP VIEW

7.2.5 WET STORE NODE

To wet store a node, click the Wet Store button. A dialog will appear showing the node serial number followed by an entry field for remarks as in Figure 65.

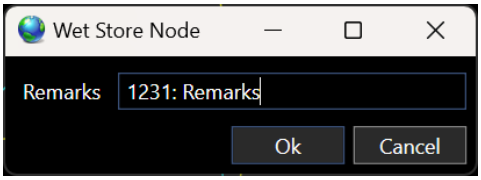


FIGURE 65 WET STORE NODE DIALOG

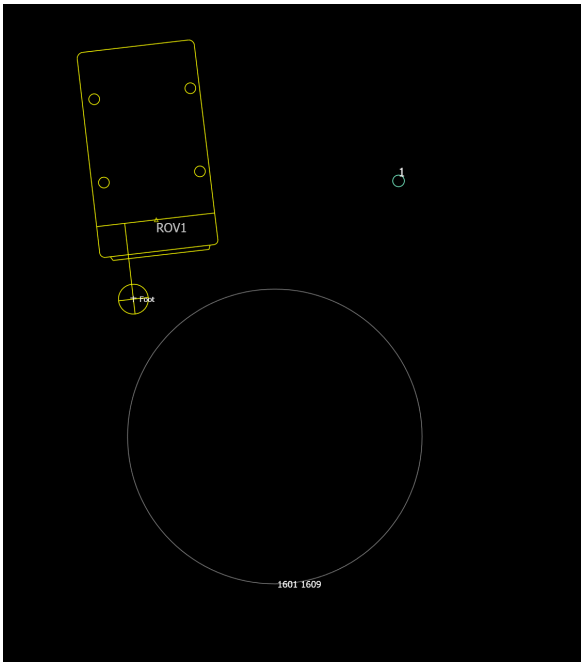


FIGURE 66 WET STORED NODE – MAP VIEW

Figure 66 shows wet stored node 1 on Map view.

Note: Wet stored node map graphics is configured in Options in the Node Dashboard Setup section. Go to Graphics Tab > Node tab.

Wet stored nodes also appear as a grid in the Wet Stored tab of the ROV Management window. If a tray is selected, the wet stored nodes show up in a light blue color, see Figure 67.

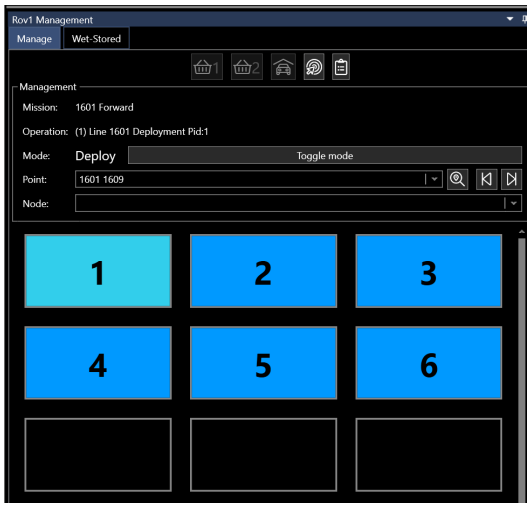


FIGURE 67 WET STORED NODE – TRAY VIEW

A node can be recovered from wet store by clicking the wet stored node in the tray view or from the wet store tab, then from the bottom part of the ROV management window click the 

button.

Once a node is retrieved it will be displayed red in the tray.

A wet stored node can also be recovered from the Map view by right clicking on the node and selecting Wet-Store from the context menu. Select the node from the menu, and then either Assign to or Recover. From Assign or Recover, select a ROV.

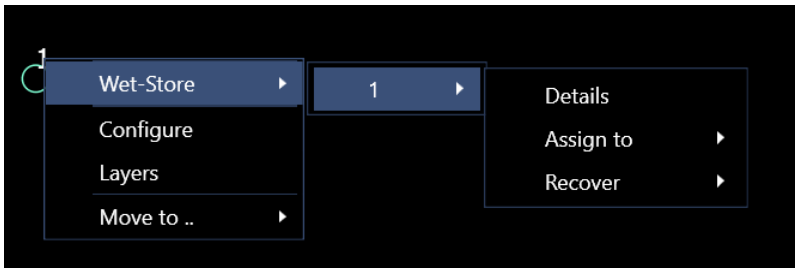


FIGURE 68 WET STORE NODE RECOVERY – MAP VIEW

If **Assign to** has been selected, the wet-store node details page will appear in the ROV management view for the respective ROV, ready to confirm and recover to that ROV's tray. If **Recover** has been selected, then the wet-stored node is recovered directly to the selected ROV's tray.

7.3 FIX MODELS

This section describes the process of taking fixes using the available fix models. Fixes are initiated when Deploy Node or Retrieve Node button is selected. The Fix Model type is assigned in the ROV Setup Node Dashboard ribbon, see Section 3.1.1.1

7.3.1 STATIC MODEL

The Static Fix is an average fix. The ROV would be stationary during this fixing operation.

FIGURE 69 STATIC FIX DISPLAY

- **Source:** The Source dropdown lists all data sources that have been setup for this fix. The primary position source is selected by default. To view data from other sources, select it in the Source drop-down

7.3.1.1 DATA VIEW

- **Details Tab:** Details Tab shows detailed capture information for the source selected in the Source Drop down. If a position source is selected, a 2D Scatter plot is displayed, along with an XY Data listing, time series for both X and Y, and a histogram for X and Y. If another data source such as Elevation is selected, then a time series, data listing and single histogram are shown for that item
- **All Data Tab:** The all data tab shows a table with statics of each of the data sources, see Figure 70

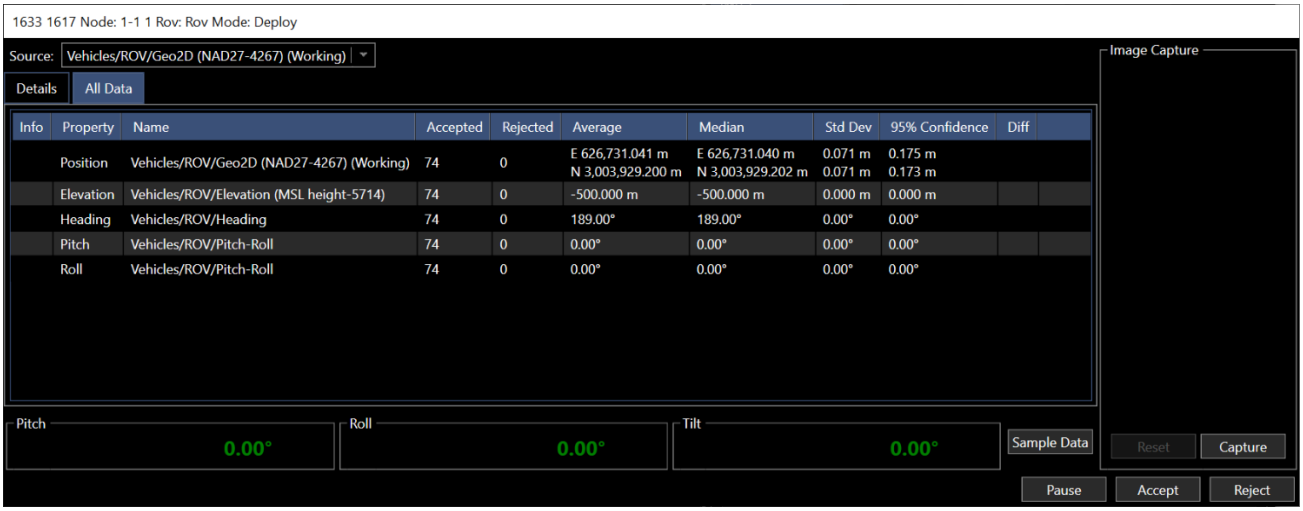


FIGURE 70 ALL DATA TAB - DATA STATISTICS

- **Node Attitude Panel:** The Node Attitude Panel shows the currently captured Pitch, Roll and Tilt for the node. The values on the left indicate the current instantaneous reading from the Node Attitude sensor. To capture a value, click the **Sample Data** button. If there is no Node Attitude Source, a default value of 0 will be applied for the Node pitch and roll

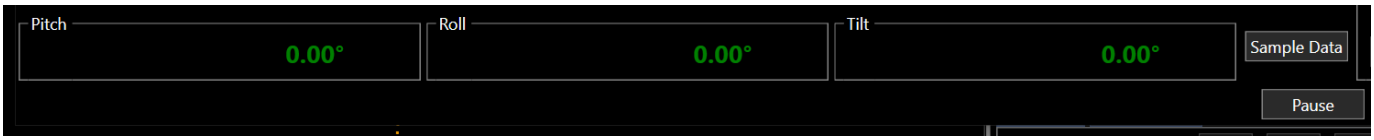


FIGURE 71 NODE ATTITUDE PANEL

- **Image Capture:** If a video source has been assigned to the ROV, then the **Image Capture** window will be displayed. Here, a capture of the video feed can be associated with the fix. Click the Capture button to grab the image currently shown in the view. This will freeze the view to that image. Click the Reset button to discard the currently capture image so that another one can be captured
- **Pause/Resume:** When enough samples have been captured, click the Pause button to stop collecting data. This will keep the fix window open and not finalize the fix. Editing can be performed such as rejecting outliers prior to accepting the fix. Once paused, the pause button changes to a Resume button. This will continue data collection so that more data can be added to the same fix
- **Accept:** When the fix is satisfactory, click the **Accept** button. This will store the results in the node fixes database and send a message to the recorder system if one has been configured

Note: A fix cannot be accepted until the **Sample Data** has been clicked.

- **Reject:** The **Reject** button will cancel the fix, close the fix window and allow the user to start over with the same node and preplot selected

7.3.2 DYNAMIC

The Dynamic Fix model is intended for situations where nodes are deployed without the ROV needing to stop. For a Dynamic position fix, the math model requires that data be collected both before and after the node is deployed. The user must start the fix well before the node is deployed and end the fix well after the node has been deployed.

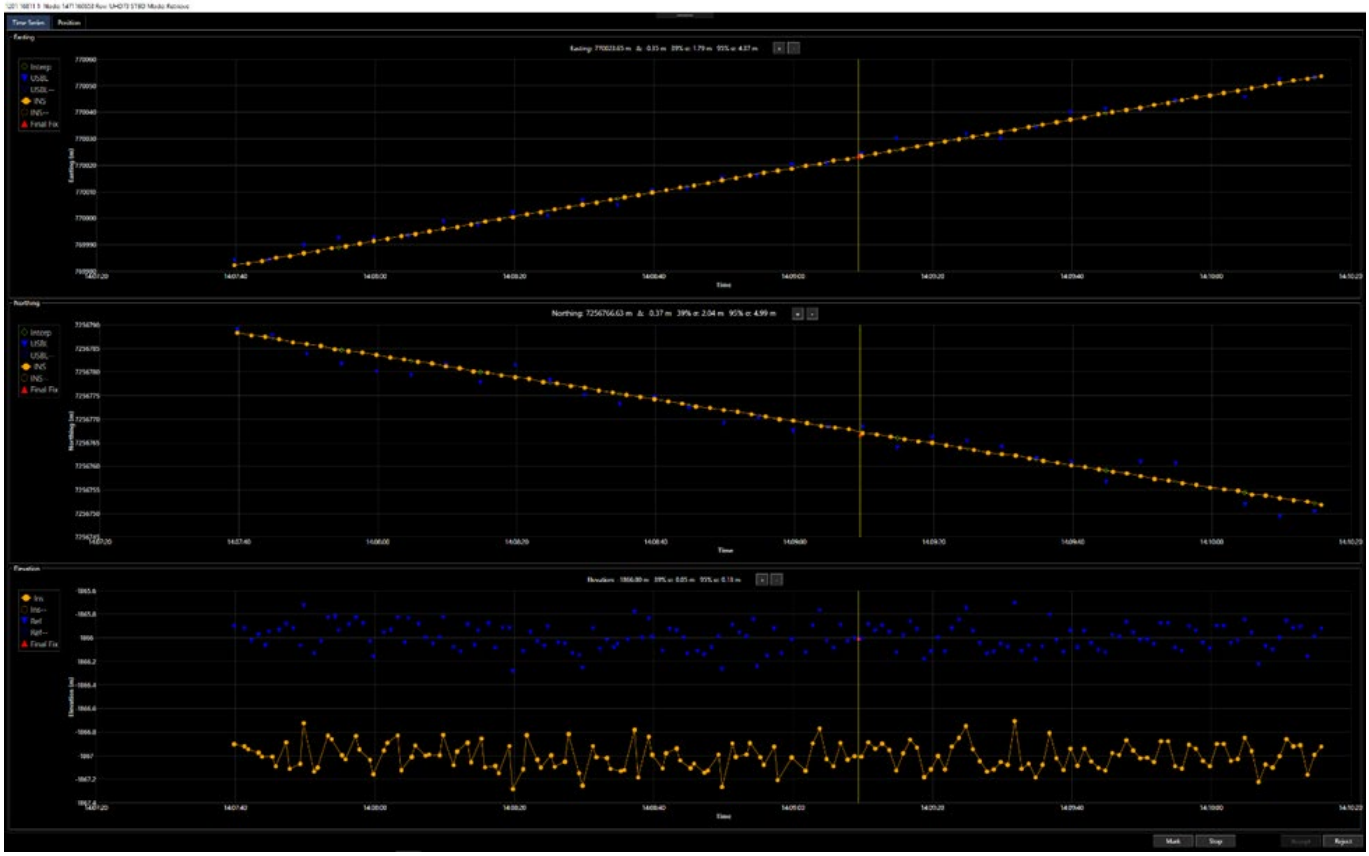


FIGURE 72 DYNAMIC FIX DISPLAY WITH MARK

7.3.2.1 TIME SERIES TAB

The Orange scatter line represents an INS path. The blue triangles represent USBL updates. The green diamonds represent the interpolated INS observations at the exact time of the USBL observation. The difference between the INS path and the USBL observation is the residual and used to adjust the final fix position.

When the node has been deployed the user **MUST PRESS** the **Mark** button. This records the timestamp at which the node was deployed. The timestamp (time-of-fix) is displayed on the time series as a vertical yellow line. As shown in Figure 72.

When the Dynamic fix has been marked with a timestamp of fix the displays are updated to show the estimated final fix location along with statistical information. The user must record sufficient data after the node has been deployed to obtain a valid fix data set.

Once enough data has been collected the user must press the **Stop** button. This stops the recording session.

Note: The Mark button must be pressed so the Stop button is available.

7.3.2.2 POSITION TAB

The Position tab displays the 2D position plot of the dynamic fix along with the pre-plot position (P) deployed position (D) and retrieved position (R) if available. Along with a chart of the residuals in easting and northing.

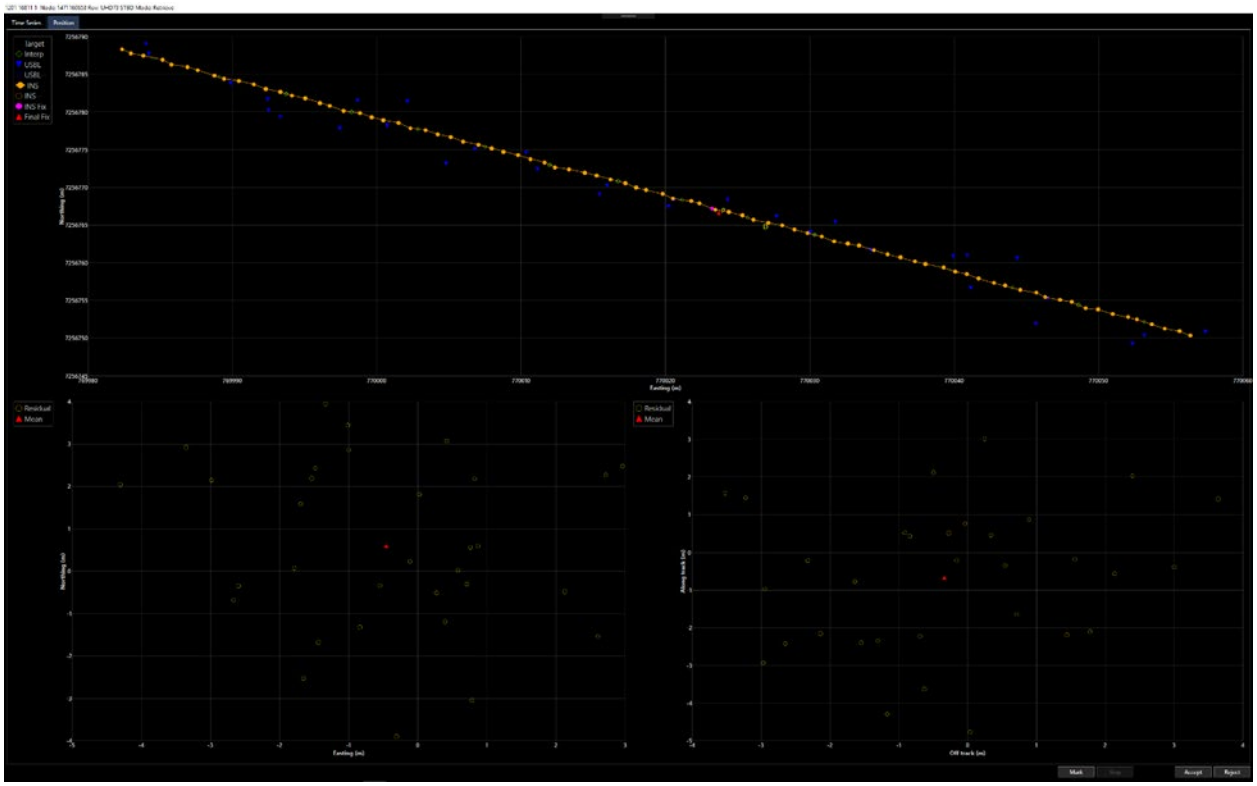


FIGURE 73 DYNAMIC FIX – POSITIONS (P, D AND R)

Operation	With Keyboard	With Mouse
Pan	Arrow keys	Shift + dragging
Zoom In	Page Up key	Scrolling down the mouse wheel
Zoom Out	Page Down key	Scrolling up the mouse wheel
Fit To Chart Plot Area	Home key	
Area Zoom		Dragging within the chart plot area selects area of the chart to zoom in.

FIGURE 74 DYNAMIC FIX VIEW - CHART NAVIGATION

If enough data has been collected, the time of the INS and USBL is properly synchronized, and the vehicle offsets are configured correctly the residuals should show a normal distribution close to 0.0.

7.3.2.3 DATA EDIT

When recording has been stopped the user may edit the data, if necessary, by rejecting obvious outliers from the data set in each of the time series displays and the 2D position plot.

The user can accept or reject individual observations in the Time Series and Position charts by selecting the individual points with the mouse.

To reject an entire region of points, highlight the region with the left mouse button while pressing CTRL key. To accept an entire region of points, highlight the region with the left mouse button while pressing ALT key.

When the data has been inspected the user can click the Accept button to accept the fix as the as-laid position of the node or click the Reject button, **ALL DATA IS DISCARDED**.

7.3.3 SNAPSHOT

When the Deploy or Retrieve button is pressed, the fix is captured. This view shows a summary of the results and allows the user to Accept or Reject. If reject is selected, the data is not stored.

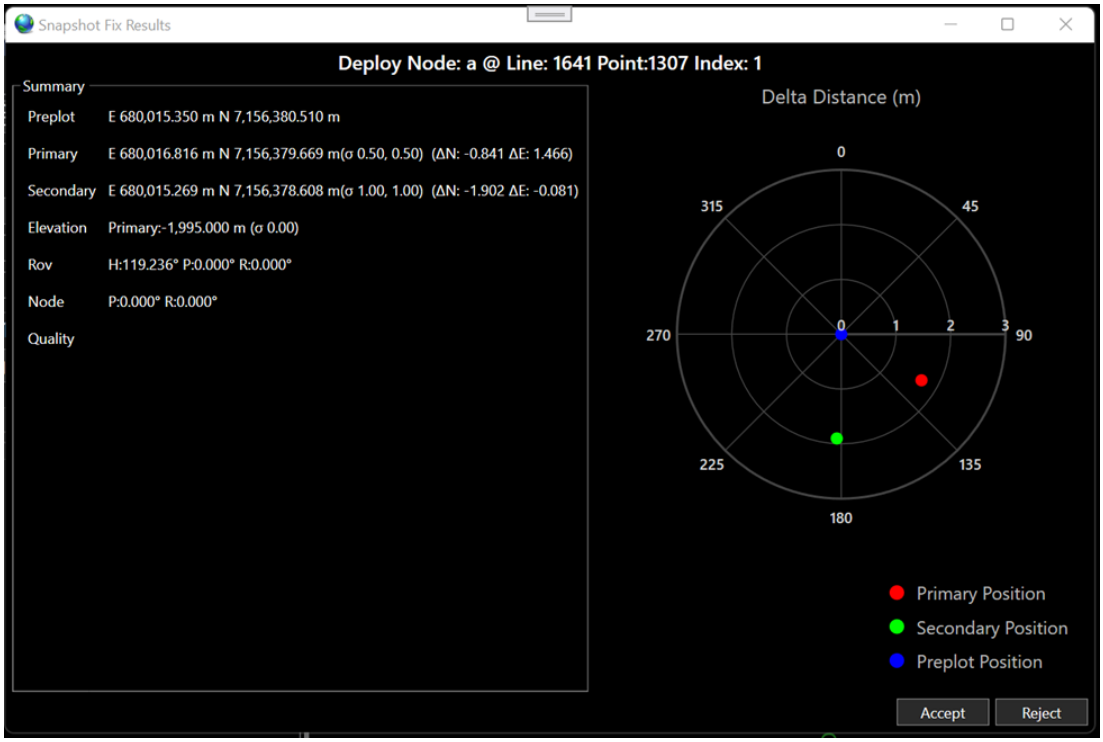


FIGURE 75 SNAPSHOT FIX

8 AS-LAID HISTORY

8.1 LIVE AS-LAID HISTORY

Live As-Laid History displays real time node activity. This is accessed by selecting History (Live) in the View section of the Node Dashboard ribbon.

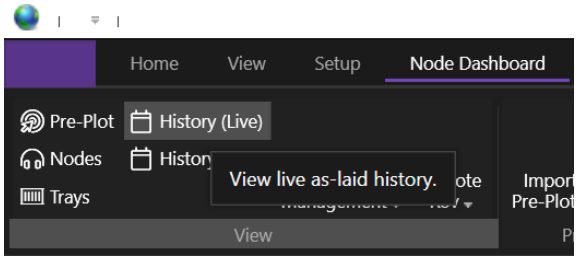


FIGURE 76 HISTORY (LIVE) – NODE DASHBOARD RIBBON

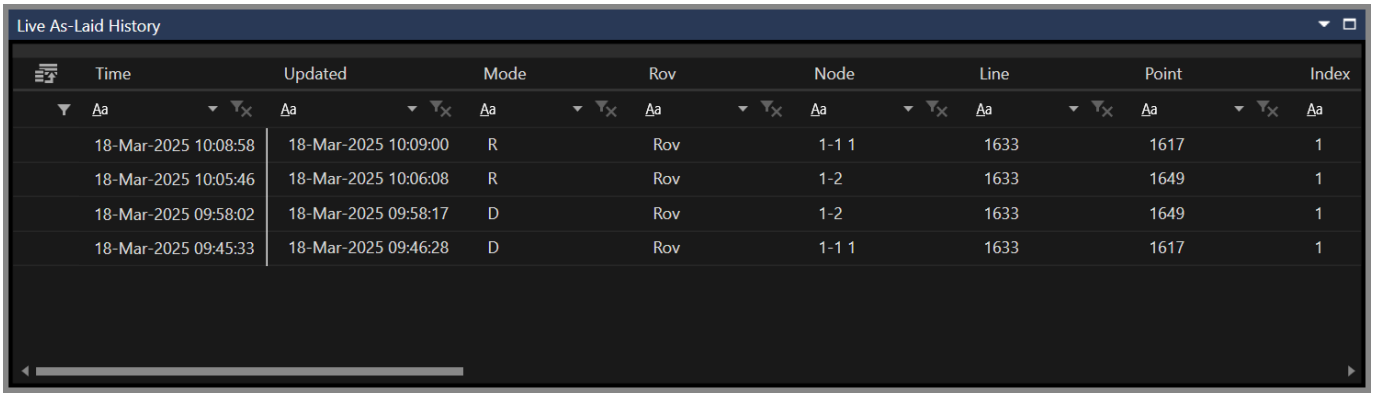


FIGURE 77 LIVE AS-LAID HISTORY DISPLAY

Specific history details can be accessed by right clicking on an item to open the context menu. Options are View Details and Add Comment.

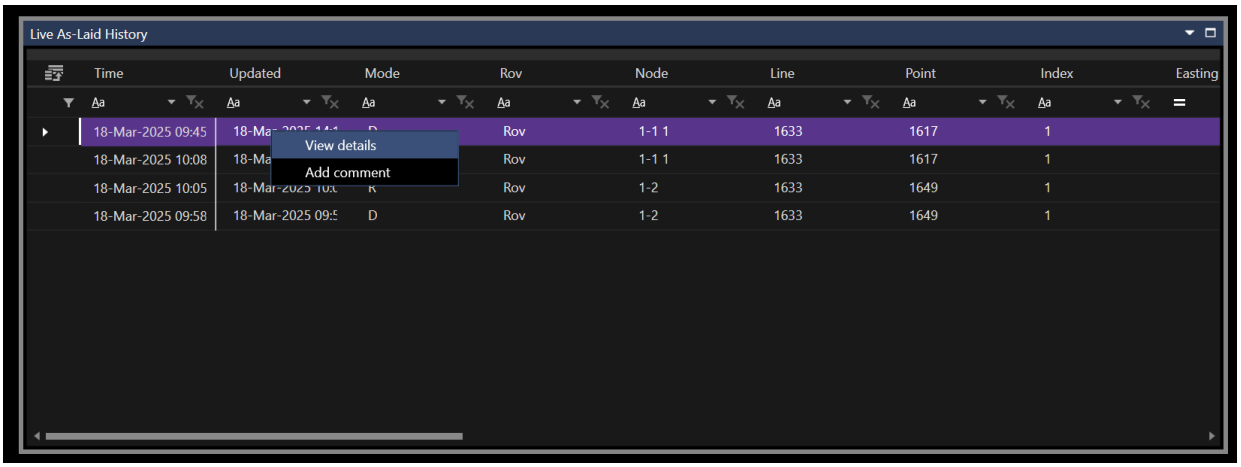


FIGURE 78 VIEW DETAILS SELECTION – LIVE AS-LAID HISTORY

Note: If any field in the as laid history exceeds the threshold parameters set in the options page, the Time field will have a red background, in addition to that field.

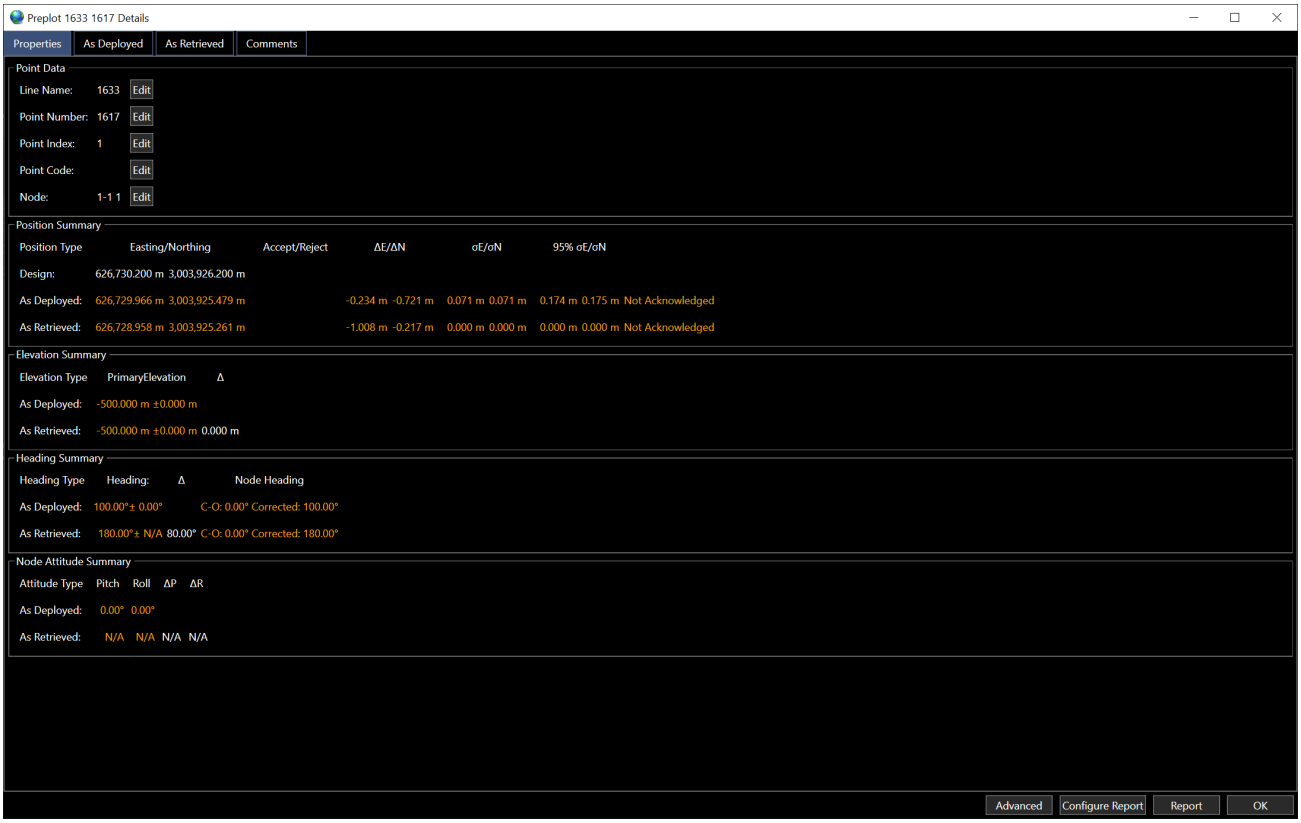


FIGURE 79 PREPLOT # DETAILS WINDOW – LIVE AS-LAID HISTORY

8.2 AS-LAID HISTORY CUSTOM QUERY

As-Laid History Custom Query allows the user to query the database for completed node activities from a user specified minimum start date and time to the current date and time This is accessed by selecting History (Full) in the View section of the Node Dashboard ribbon.

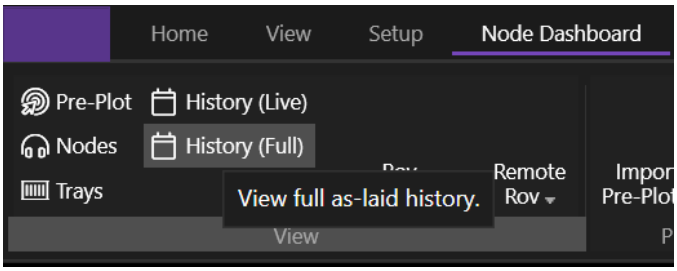


FIGURE 80 HISTORY (FULL) – NODE DASHBOARD RIBBON

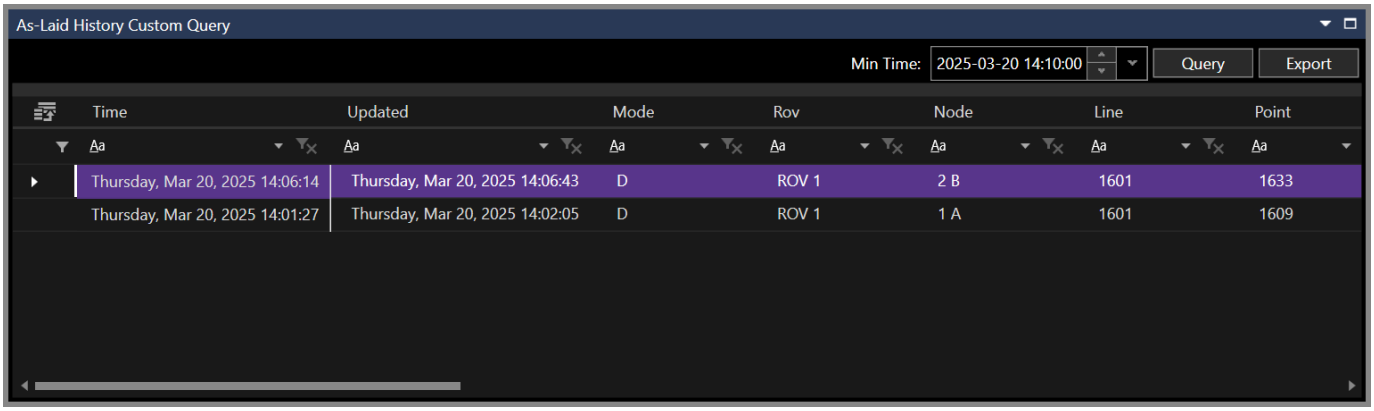


FIGURE 81 AS-LAID HISTORY CUSTOM QUERY DISPLAY

Specific history details can be accessed by right clicking on an item to open the context menu. Options are View Details and Add Comment.

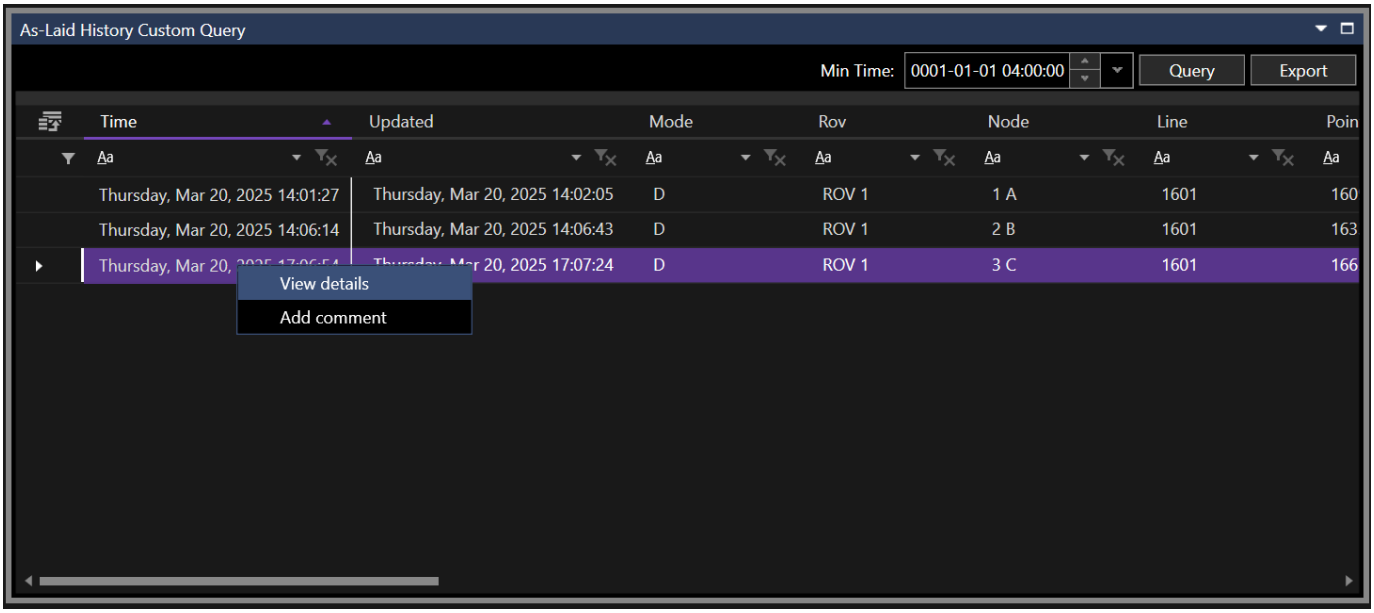


FIGURE 82 VIEW DETAILS SELECTION – AS-LAID HISTORY CUSTOM QUERY DISPLAY

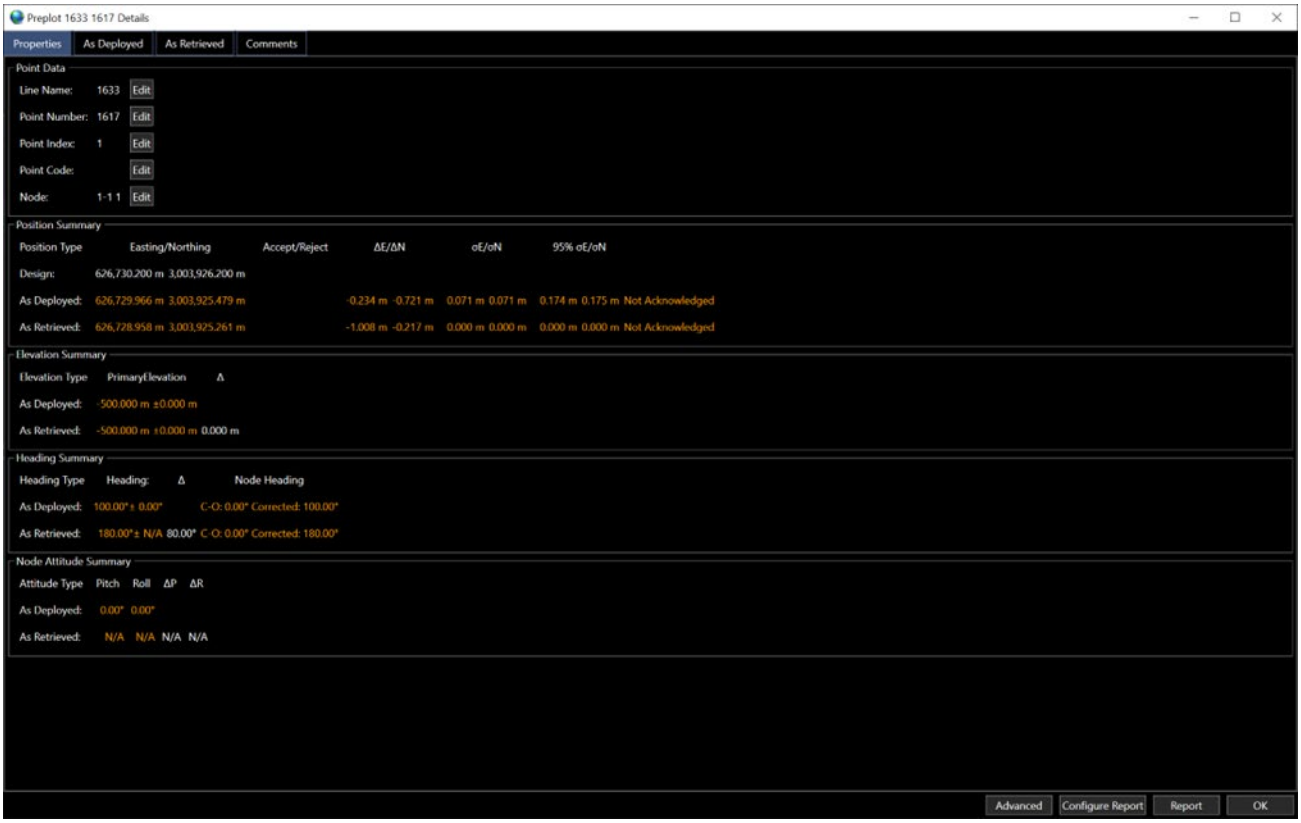


FIGURE 83 PREPLOT # DETAILS WINDOW – AS-LAID HISTORY CUSTOM QUERY

9 NODE DATA REVIEW

9.1 VIEW NODE/POINT DETAILS

To review a completed node, go to the Preplots window, Live As-laid History or As-laid History Custom Query and right-click on the completed node to open the context menu.

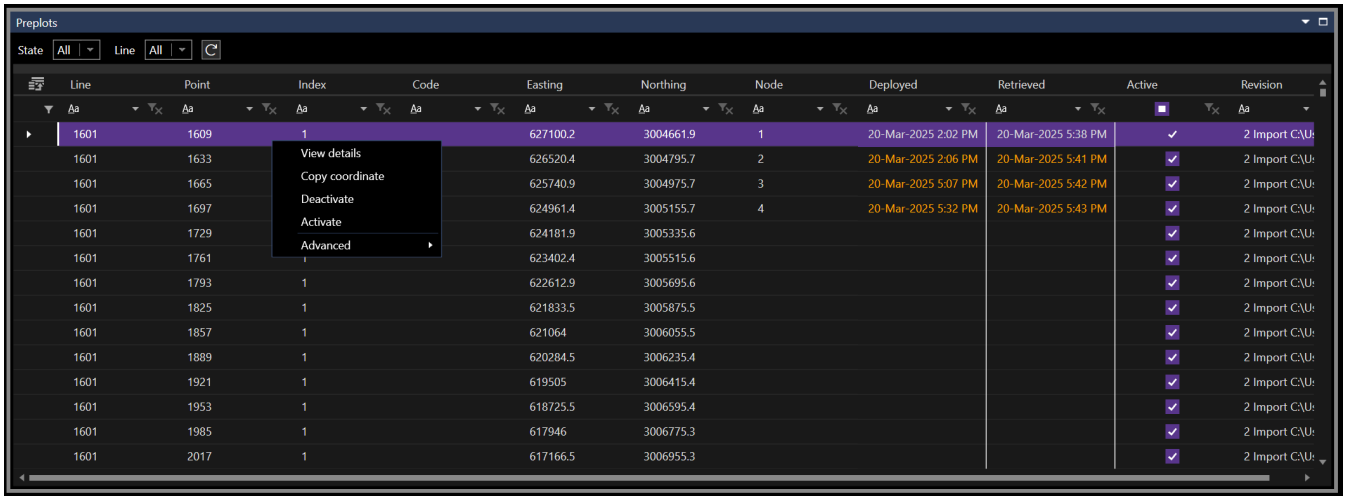


FIGURE 84 PREPLOTS WINDOW

Note: The time display (UTC or Local) is controlled in project preferences.

Selecting **View details** opens the Preplot Details window displaying a summary of the selected point and node data.

9.1.1 PROPERTIES TAB

a. If a node has not been deployed at the selected point the Properties tab presents the point information including the design location (Preplot)

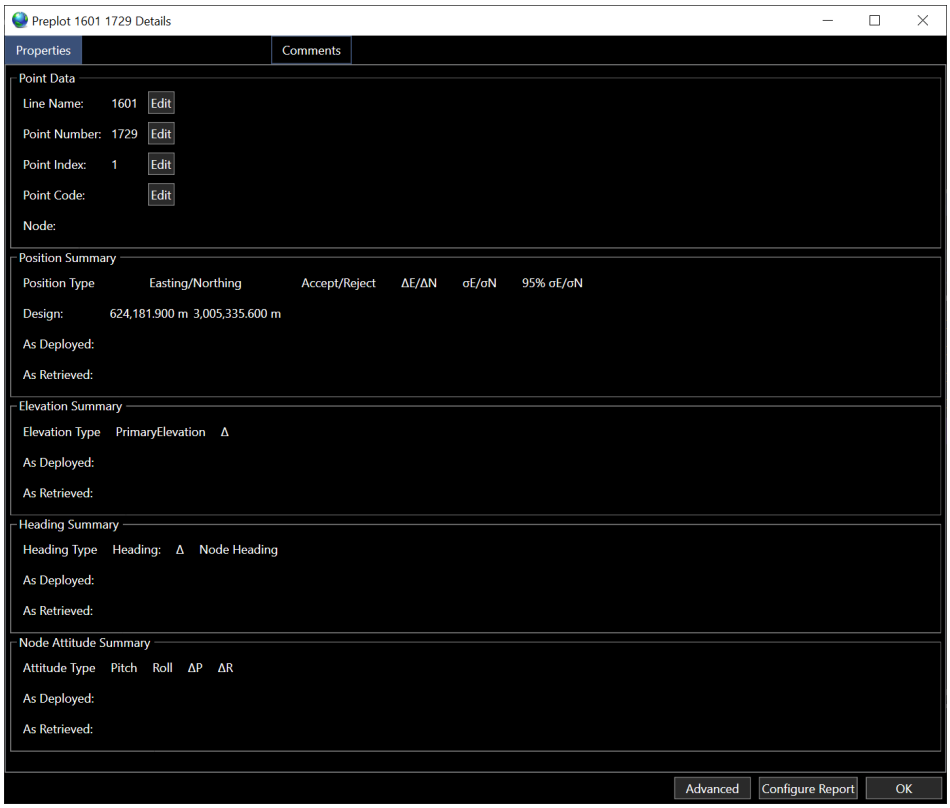


FIGURE 85 PREPLOT DETAILS WINDOW – POINT INFORMATION

b. If a node has been deployed at the selected point, the Properties tab, see Figure 86 presents the information and a summary of the **As Deployed** position

- Easting and Northing
- Number of Accepted and Rejected Fixes
- Delta Easting and Northing from the design location
- 1 sigma standard deviation of the Easting and Northing of the fix sample
- 95% confidence standard deviation of the Easting and Northing of the fix sample

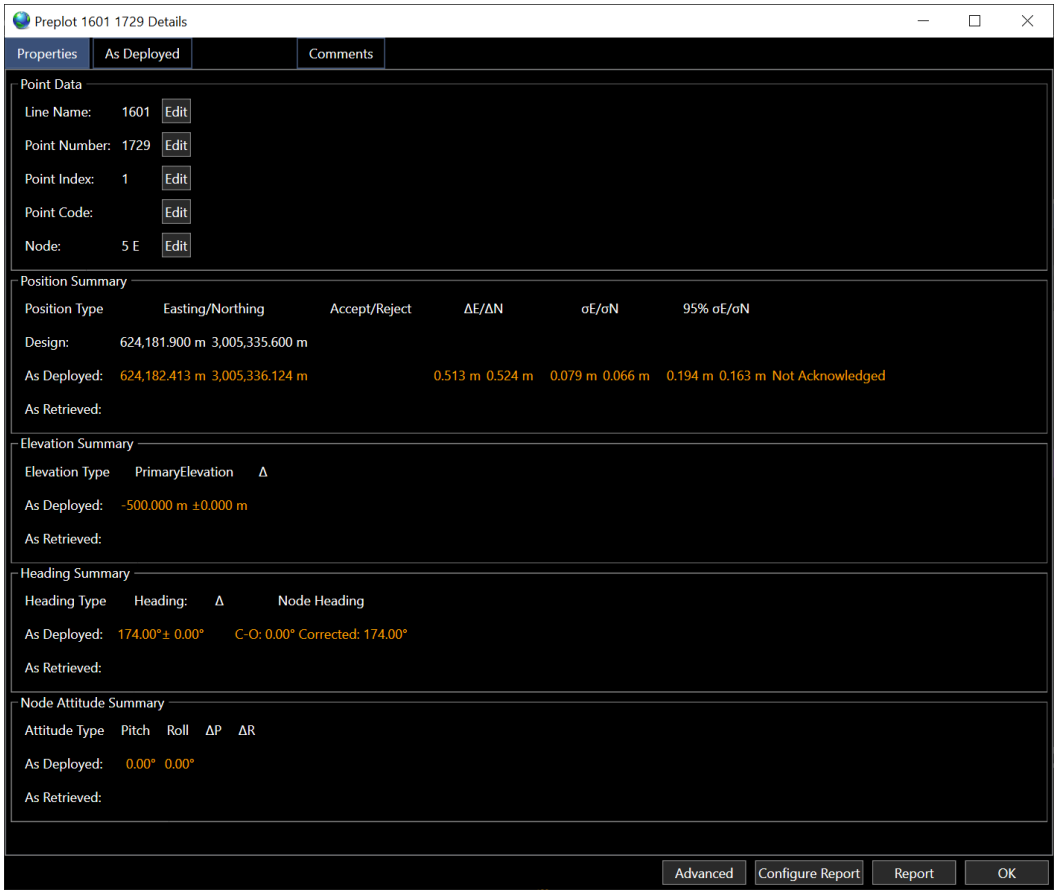


FIGURE 86 PREPLOT DETAILS WINDOW – DEPLOYED NODE INFORMATION

c. If a node has been retrieved from the selected point the Properties tab, see Figure 87, presents the information and a summary of the **As Deployed** and **As Retrieved** position.

Note: The delta Easting and delta northing for the retrieved position is from the deployment to the retrieval not to the target location.

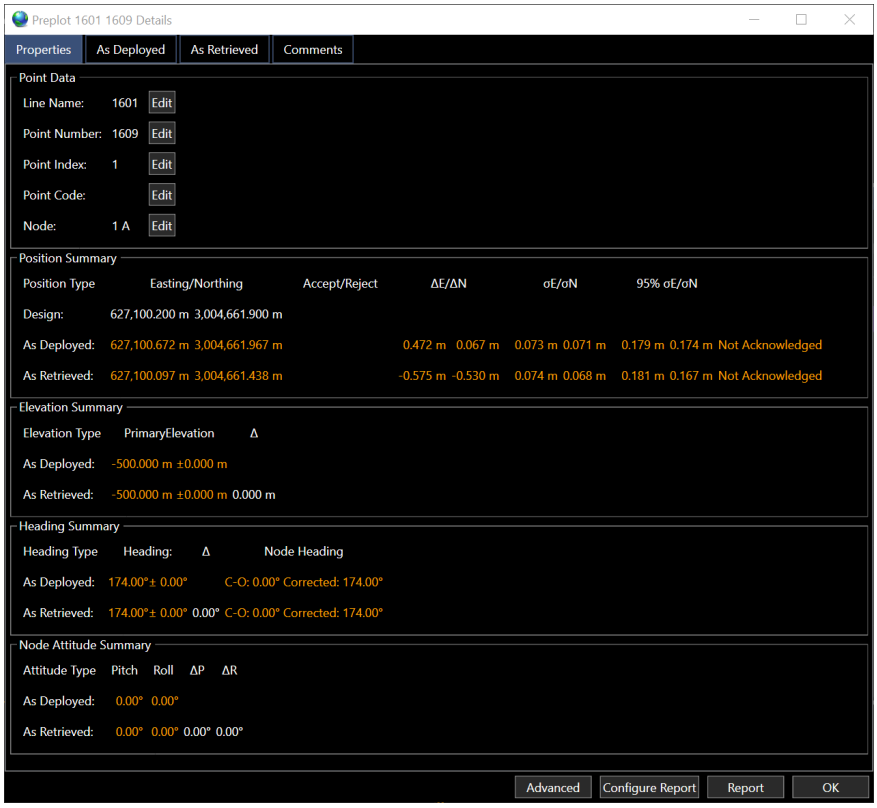


FIGURE 87 PREPLOT DETAILS WINDOW – DEPLOYED/RETRIEVED NODE INFORMATION

9.2 VIEW/EDIT NODE FIXES

Full details of a node deployment/retrieval can be viewed and edited from the **As Deployed** or **As Retrieved** tabs in the pre-plot details window.

9.2.1 SUMMARY TAB

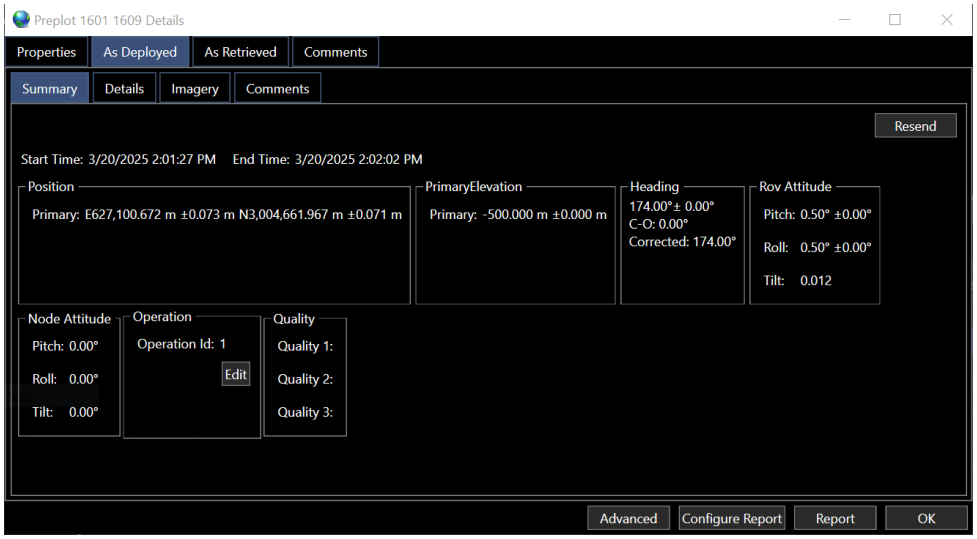


FIGURE 88 AS DEPLOYED – SUMMARY TAB

- **Position:** The primary position in easting and northing with standard deviation is shown. If a secondary position data source is available, the secondary value is shown as well, and compared to the primary
- **Elevation:** The primary elevation data source is shown with standard deviation. If a secondary data source is available as well, this is shown and compared to the primary source
- **Heading:** The ROV heading data source result and standard deviation are displayed
- **ROV Attitude:** The pitch roll and tilt of the ROV are displayed with average values
- **Node Attitude:** The node attitude values are displayed with tilt
- **Quality:** Logged numerical data for reporting purpose

If the connection between Node Dashboard, and the recorder was not operable when the node fix was completed. The view will indicate that the positions have not been acknowledged by displaying the date and time for As Deployed and Retrieved **orange**.

Transmission can be sent again using the **Resend** button. If the raw data of the fix is edited, such as outliers rejected, the Resend button can also be used to send the updated result.

9.2.2 DETAILS TAB

The Details tab shows the raw data used for each of the position sources that are averaged. That is all data sources except for the Node attitude, which is instantaneously grabbed and recorded.

9.2.2.1 DETAILS XY SCATTER

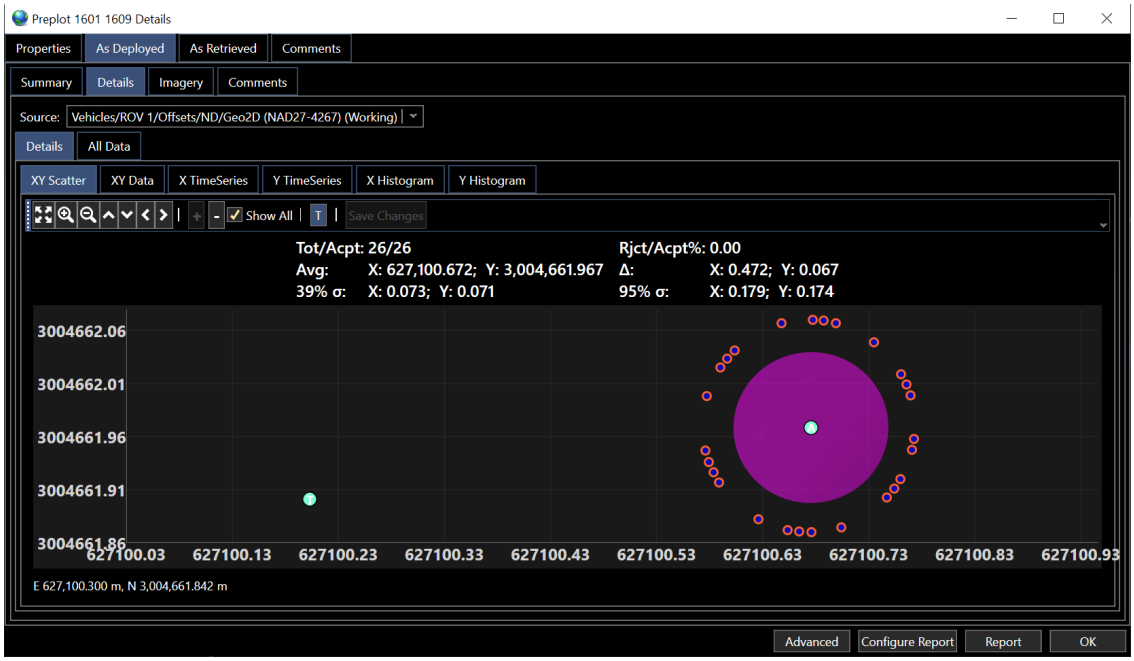


FIGURE 89 AS DEPLOYED – DETAILS 2D SCATTER PLOT

The position sources show a 2D scatter plot in easting and northing, when selected, as displayed in Figure 89.

- Values included in the fix are displayed as circles
- Values excluded are displayed as triangles
- The target value and average value are included as circles with the letter T and A respectively

The remove and accept data buttons  are used for the following:

a. Clicking the '-' button automatic remove button will remove the furthest outlier compared to the average
b. Clicking the '+' automatic add button will accept the closest outlier compared to the average
c. For non-position source, data is represented by a time series and histogram. The Histogram is read only, and gives an indication of data quality, while the Time Series can be edited
d. In the time series view, the accepted data shows up as squares, while rejected data is represented as X's. Click on a value to reject it
e. To accept multiple items, hold down the 'a' key on the keyboard and drag a box around them items

f. To reject multiple items, hold down the ‘r’ key on the keyboard, and a drag a box around them

9.2.2.2 ALL DATA

- **Selecting a Data Source:** To view a data source, select it from the list and the details will appear in the lower panel
- **All Data Details:** In the main list view, for each data source, the average, median, standard deviation, and 95% error are shown

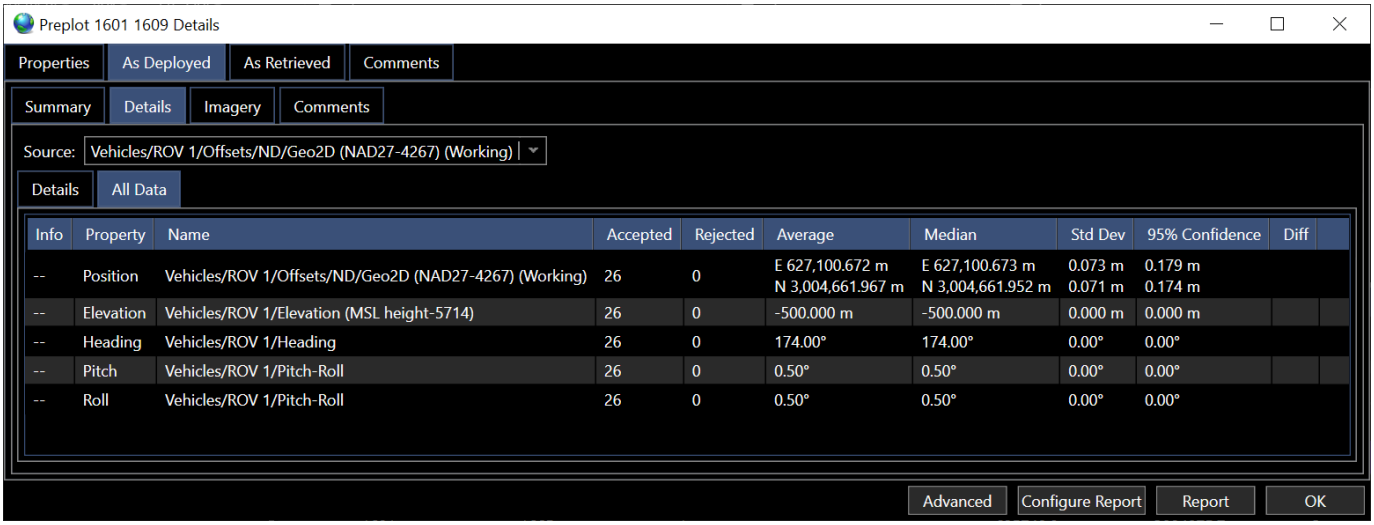


FIGURE 90 AS DEPLOYED – ALL DATA

The average of the primary position and the primary elevation are sent to the recorder. The median is displayed as well in this view. The median is less sensitive to outliers in the data. If the median differs greatly from the average, review the raw data, and remove any bad outliers.

9.3 GENERATE FIX REPORT

A report can be generated for an individual pre-plot which can be configured to give a one page summary of the deployment, or full details with graphs.

1. From the Pre-plot table, select the point, right mouse click and click on **View Details**, see Figure 87.
2. Click the **Configure Reports** button to open the Report Configuration dialog. The report by default includes a summary of the Preplot point and the As Deployed and As Retrieved positions if they are available.

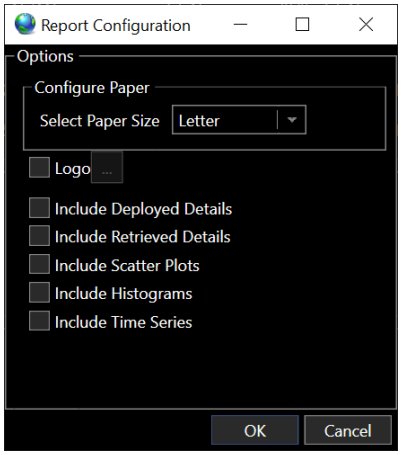


FIGURE 91 PREPLOT DETAILS - PROPERTIES TAB - CONFIGURE REPORTS DIALOG

3. Click **Report** to generate the report.
4. To export the report, from the tool bar on the bottom of the **Node Position Report** window select the desired file format and then click **Export.**

9.4 SPECIAL CASE EDITING

The Preplot Details view supports editing of logged data to correct any errors that were made during a deployment/retrieval. The Editing buttons are shown below in Figure 92.

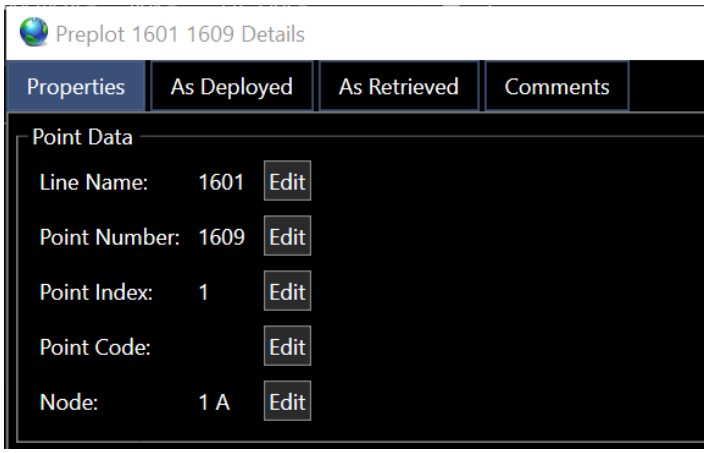


FIGURE 92 SPECIAL CASE EDITING OPTIONS – POINT DATA

Note: All special case editing options require the Administrator user role.

9.4.1 CHANGE THE LINE

To change the Line Name, click the Edit button associated with the Line Name option.

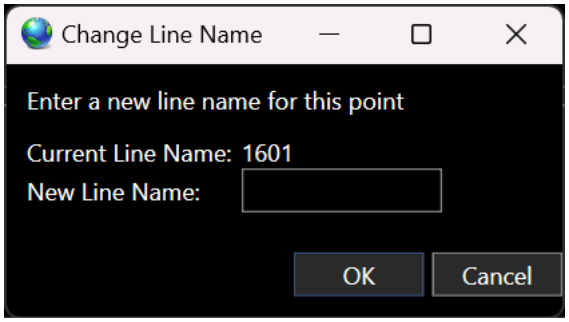


FIGURE 93 POINT DATA - EDIT LINE NAME

The currently selected line name is shown. To change the line name, type the new value, and click ok.

Note: This will update the value in the NodeDashboard database but will not send the value to the recorder system. To resend the message to the recorder system with the new line name, the **Resend** option can be used on either the deployed or retrieved page, see section 9.2.1

9.4.2 CHANGE THE POINT NUMBER

To change the point number saved in the database for a preplot, click the Edit button associated with the Point Number option.

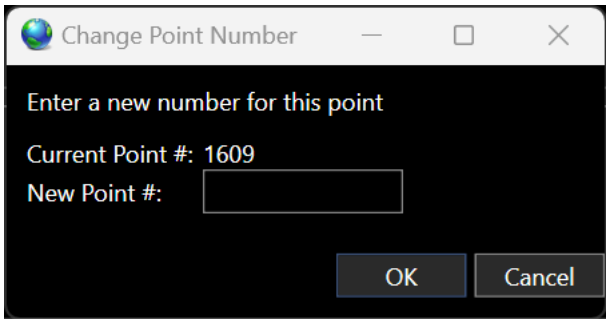


FIGURE 94 POINT DATA - EDIT POINT NUMBER

The current point # is shown. To change the point #, type the new value, and click ok.

Note: This will update the value in the NodeDashboard database but will not send the value to the recorder system. To resend the message to the recorder system with the new preplot point number, the **Resend** option can be used on either the deployed or retrieved page, see section 9.2.1

9.4.3 CHANGE THE POINT INDEX

To change the point index associated with a preplot, click the Edit button associated with the Point Index on the details page.

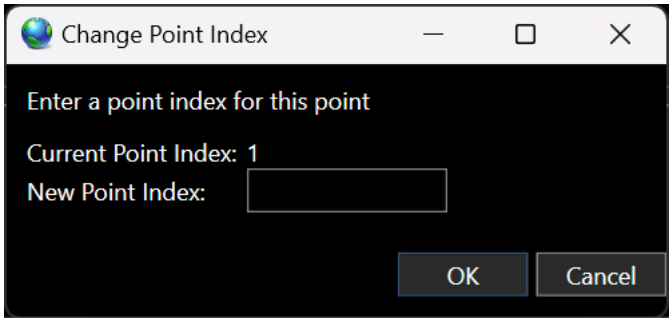


FIGURE 95 POINT DATA - CHANGE POINT INDEX

Type the new index in the text box and click Ok to save the changes to the database.

Note: This will update the value in the NodeDashboard database but will not send the value to the recorder system. To resend the message to the recorder system with the new preplot point number, the **Resend** option can be used on either the deployed or retrieved page, see section 9.2.1

9.4.4 CHANGE THE POINT CODE

To change the point code associated with a preplot, click the Edit button associated with the Point Code on the details page.

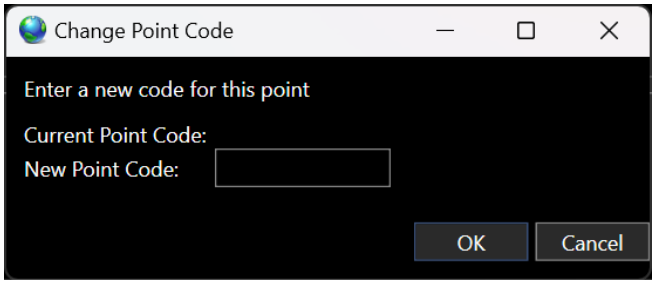


FIGURE 96 POINT DATA - CHANGE POINT CODE

Type the new point code value into the text box and click Ok to save the changes to the database.

Note: This will update the value in the NodeDashboard database but will not send the value to the recorder system. To resend the message to the recorder system with the new preplot point number, the **Resend** option can be used on either the deployed or retrieved page, see section 9.2.1

9.4.5 EDIT THE NODE

The Edit node allows the user to choose a new node from a list using the drop-down button.

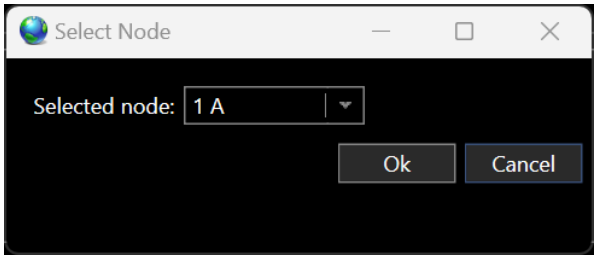


FIGURE 97 POIT DATA - SELECT NODE DIALOG

The **Select Node** opens a dialog box with the list of nodes existing in the project which are not deployed at other locations. To change the node, select a node from this list and click OK.

Note: This will update the value in the NodeDashboard database but will not send the value to the recorder system. To resend the message to the recorder system with the new node, the **Resend** option can be used on either the deployed or retrieved page, see section 9.2.1

9.5 REMOVING DEPLOYMENT AND RETRIEVALS FROM A PREPLOT

If the wrong location was chosen in NodeDashboard for deployment and a deployment was completed, this option allows for that deployment to be removed from the preplot. This option may also be useful for testing/training purposes on the system, where a preplot can be used for a practice deployment and then that deployment removed.

To access the removal options, click the Advanced button at the bottom of the Preplot Details view. This will open the Edit Preplot Aslaids dialog, see Figure 99.

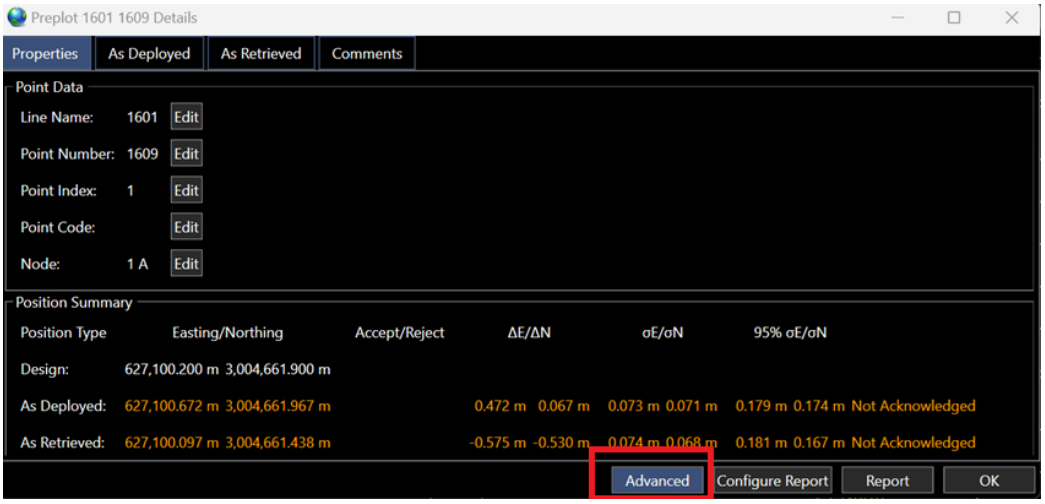


FIGURE 98 PREPLOT DETAILS – ADVANCE BUTTON

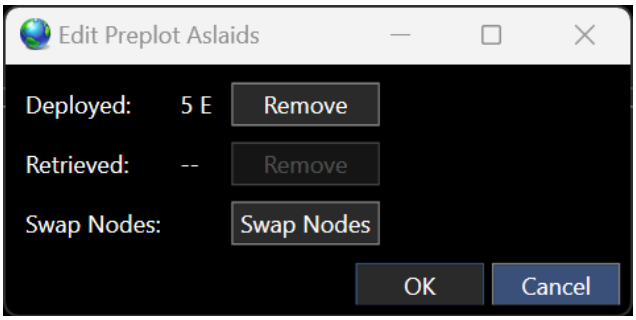


FIGURE 99 EDIT PREPLOTS ASLAIDS DIALOG

If a node has been deployed at the preplot location, the Remove button will be highlighted. Likewise, if a node has been subsequently retrieved from the location, the Remove button associated with the retrieved option will be enabled.

NodeDashboard will prompt the user to confirm any changes. Once the changes are confirmed, the database is edited such that the link is removed from the deployment/retrieval details to the preplot, but the data is not removed from the database.

9.5.1 REMOVING A DEPLOYMENT

If a deployment is present at the preplot location, the associated node will either have Deployed Status or Retrieved Status. When the deployment is removed, the node will be reset to Deployable status, such that it is ready to place at another location.

Click the **Remove** button next to deployment.

9.5.2 REMOVING A RETRIEVAL

When the retrieval is removed, no change is made to the node. The retrieved node would have been set to Retrieved status. It is already possible to deploy retrieved nodes at any location, so no change is required to the node.

Click the **Remove** button next to retrieval.

9.6 PREPLOT COMMENTS

Preplot comments can be used to save any pertinent text information about a location. Preplot comments are accessed by clicking on the Comments tab in the Preplot Details view.

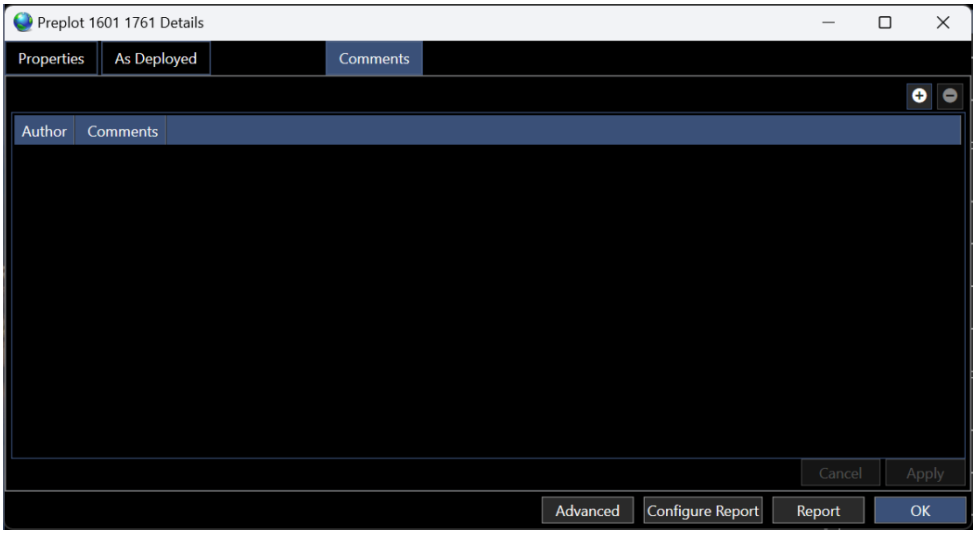


FIGURE 100 PREPLOT COMMENTS VIEW

1. To add a new comment, click the  button on the Comments tab.

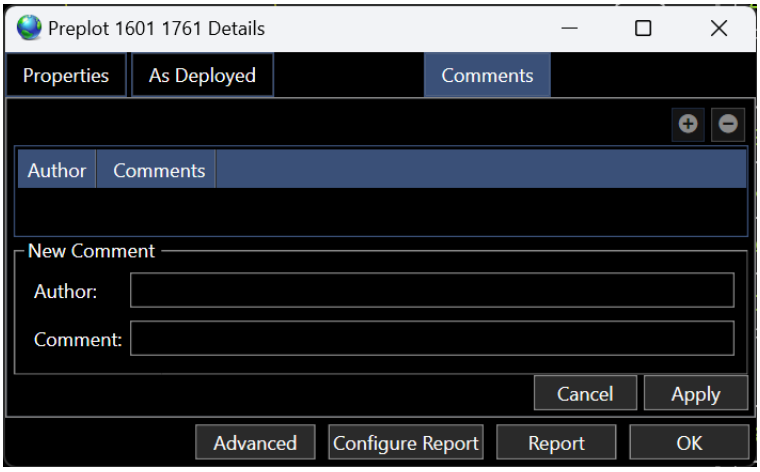


FIGURE 101 ADDING A COMMENT

2. Type the Author name and the comment. Click **Apply** to save or **Cancel** to discard the comment. The comment is then saved in the database associated with the preplot.
3. To remove a comment, select one in the list and click the  button. Administrator privileges are required to remove a comment.

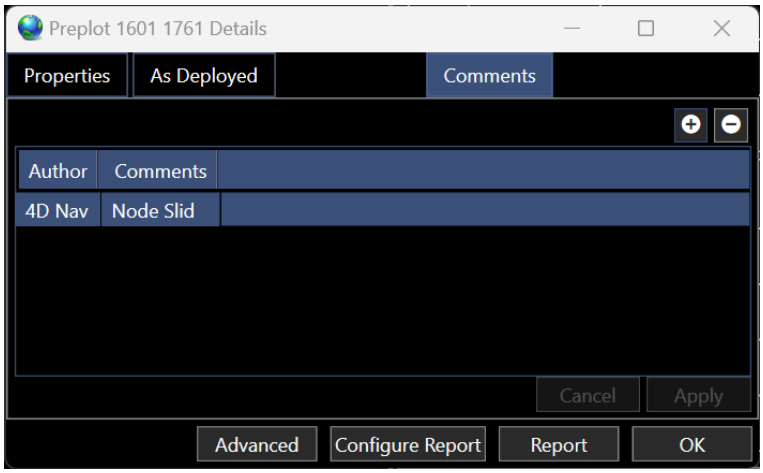


FIGURE 102 REMOVE A COMMENT

Comments are shown in the list view in the comments tab of the Preplot Details view. Comments are also included in the detailed summary report for export as deliverables.

10 REPORTS

10.1 ACTIVITY SUMMARY REPORT

The Activity Summary Report enables the user to generate a summary snapshot of activities by specifying a Time Filter, Data Filter and/or State Filter. It provides a count for the items completed and a Nodes/Hour record. This can be used for a daily report.

1. To create an Activity Summary Report, Click the button **“Activity Summary Report”** on the **Node Dashboard** tab. This will open the Activity Summary Report dialog.

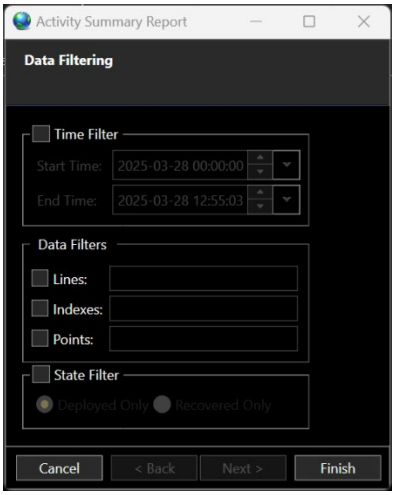


FIGURE 103 ACTIVITY SUMMARY REPORT DIALOG

2. Specify a time period for which to report on by enabling the Time Filter. The **End Time** defaults to the current time, the **Start Time** is start of the current day. The UTC/Local option is set in the project configuration.

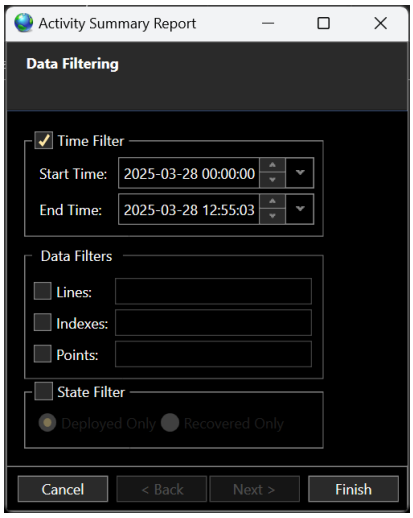


FIGURE 104 ACTIVITY SUMMARY REPORT DIALOG – TIME FILTER

3. Additional filters can be selected to customize report. These include **Data Filters** and **State Filter**.

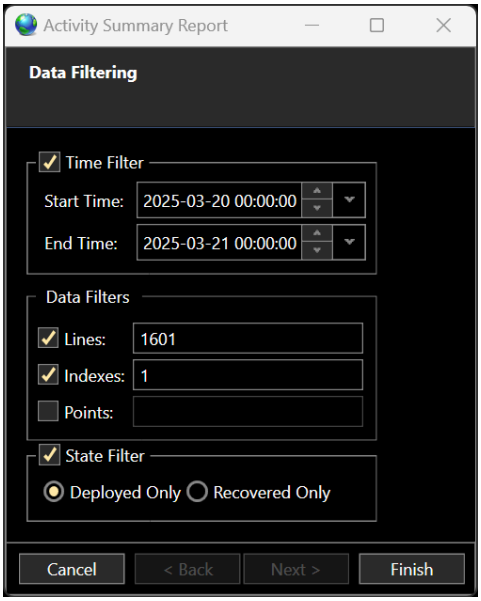


FIGURE 105 ACTIVITY SUMMARY REPORT DIALOG – FILTERS ENABLED

4. Click Finish to create the report.

Activity Summary from Thursday, Mar 20, 2025 00:00:00 to Friday, Mar 21, 2025 00:00:00 (Utc)

#	Time	Activity	Line	Station	Node	ROV
1	Thursday, Mar 20, 2025 14:02:02	Deployed	1601	1609	1 A	ROV 1
2	Thursday, Mar 20, 2025 14:06:41	Deployed	1601	1633	2 B	ROV 1
3	Thursday, Mar 20, 2025 17:07:22	Deployed	1601	1665	3 C	ROV 1
4	Thursday, Mar 20, 2025 17:32:00	Deployed	1601	1697	4 D	ROV 1
5	Thursday, Mar 20, 2025 17:38:32	Retrieved	1601	1609	1 A	ROV 1
6	Thursday, Mar 20, 2025 17:41:17	Retrieved	1601	1633	2 B	ROV 1
7	Thursday, Mar 20, 2025 17:42:49	Retrieved	1601	1665	3 C	ROV 1
8	Thursday, Mar 20, 2025 17:43:59	Retrieved	1601	1697	4 D	ROV 1
	Deployed	Retrieved	Combined	Nodes/hr		
Total	4	4	8	0.4		

FIGURE 106 ACTIVITY SUMMARY REPORT EXAMPLE

5. The report can be exported as PDF or Excel 2007.

10.2 DETAILED SUMMARY REPORT

The detailed summary report provides a data grid of all pre-plots, with full deployment/retrieval details and comparisons. This can be exported as a comma separated values (csv) file for import into Excel or other spreadsheet software.

The user can filter data to be exported by selecting a Time Filter, Data Filters or State. The configuration is the same as the Activity Summary Report above.

1. To create a Detailed Summary Report, Click the button **“Detailed Summary Report”** on the **Node Dashboard** tab. This will open the Detailed Summary Report dialog.

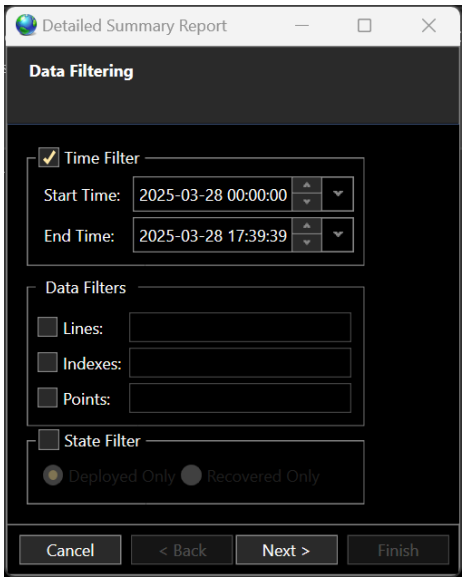


FIGURE 107 DETAILED SUMMARY REPORT – DATA FILTERING DIALOG

2. Select the filters to be used for data to be exported.
3. Click Next. To open the Report Type dialog.

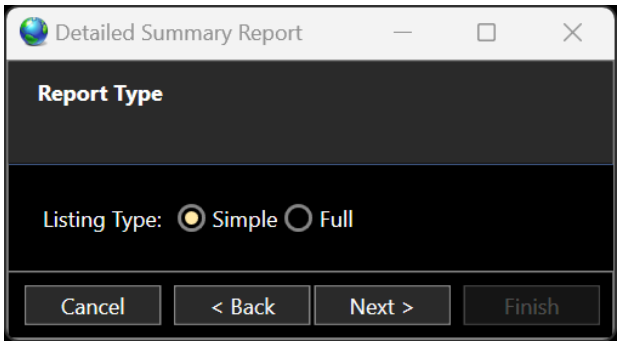


FIGURE 108 DETAILED SUMMARY REPORT DIALOG – REPORT TYPE DIALOG

- **Simple:** The simple export contains details about the deployed and/or recovered positions and compares them to the proposed preplot position
- **Full:** The full export contains additional fields, where if a primary and secondary position source is configured for the ROV, the two sources are compared

4. Click Next. To open the Position Units dialog.

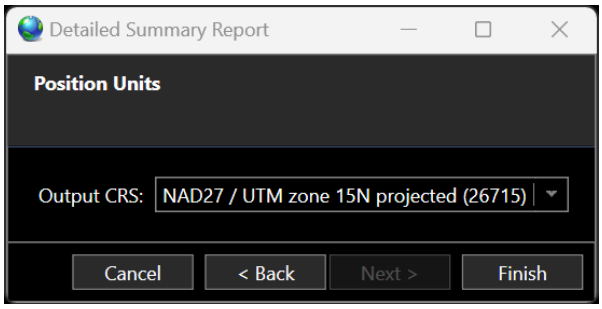


FIGURE 109 DETAILED SUMMARY REPORT DIALOG – POSITION UNITS DIALOG

5. The Position Units dialog allows the user to select what units to be used for the exported data. This is controlled by the selected horizontal CRS in the drop-down list.

Note: The Output CRS must have already been defined in the Horizontal CRS setup page found on the Setup tab of the ribbon.

10.3 EXPORTS

For more customized forms of Node data export. Data can be exported as a .CSV file type, Node information in SPS 2.1 seismic data format or a P1/11 Receiver file.

10.3.1 EXPORT RECEIVER DATA

Export Node data as a SPS 2.1 file or custom CSV file. Click the button **Export Receivers** on the Node Dashboard tab. This will open the Receiver Export Settings dialog.

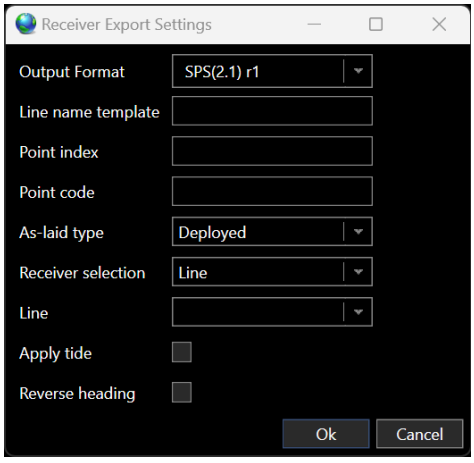


FIGURE 110 RECEIVER EXPORT SETTINGS DIALOG

- **Output Format:** From drop-down select output type
- **Line name template:**
- **Point index:** Overrides output point index with value entered. If not used leave empty
- **Point code:** Overrides output point code with value entered. If not used leave empty
- **As-Laid type:** Select Deployed or Retrieved
- **Receiver selection:** Select Line, Line Range or Time
  - **Line:** Select a single line
  - **Line Range:** Allow user to select multiple lines. Enter as i.e. 1200,1210-1220,1300
  - **Time:** Select Start time and End Time
- **Apply tide:** Enable to apply tide data. Select Tide station that has been added in Database Services (Tides). Select Tide interpolation method

Note: Refer to Tides in Database Services section of NavView User Guide.

Click Okay to save the *.r01 SPS file type or *.csv file type.

10.3.2 EXPORT SPS 2.1 RECEIVER FILE

Shell Processing Support (SPS) data format for exchanging Geophysical positioning data.

To export Node information in SPS 2.1 seismic data format, Click the button **SPS 2.1** on the Node Dashboard tab. This will open the Activity Summary Report dialog.

1. Select Data Filtering.

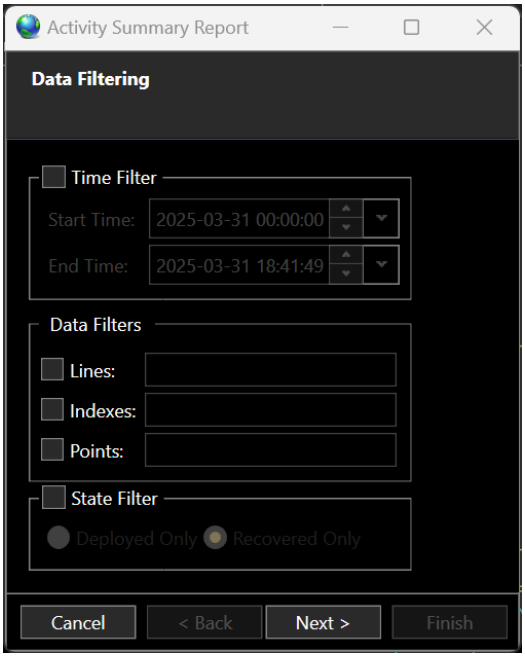


FIGURE 111 SPS 2.1 EXPORT – DATA FILTERING PAGE

2. Click Next to open the Output Selection page. Select Deployed or Recovered to export.

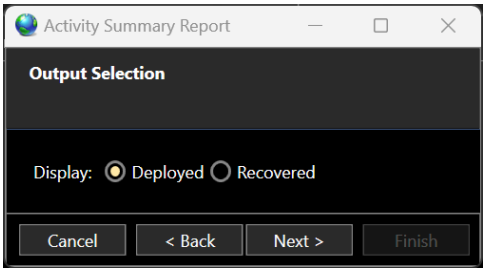


FIGURE 112 SPS 2.1 EXPORT – OUTPUT SELECTION PAGE

3. Click Next to open the Position Units page.

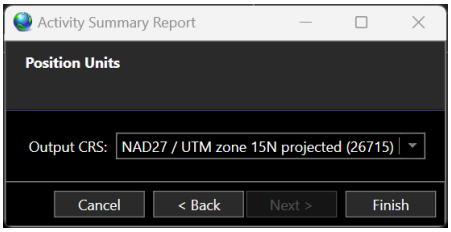


FIGURE 113 SPS 2.1 EXPORT – POSITION UNITS PAGE

4. The Position Units dialog allows the user to select what units to be used for the exported data. This is controlled by the selected horizontal CRS in the drop-down list.

Note: The Output CRS must have already been defined in the Horizontal CRS setup page found on the Setup tab of the ribbon.

5. Click Finish, this will open the SPS Export window.

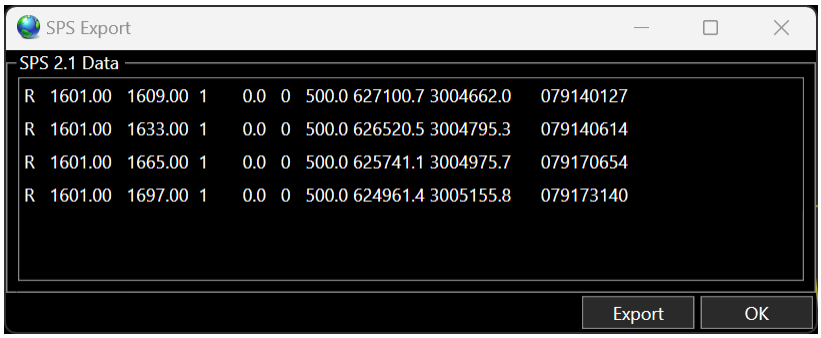


FIGURE 114 SPS EXPORT PAGE

6. Click Export to save the (*.R01) file.

10.3.3 EXPORT P1/11 RECEIVER FILE

The OGP P1/11 is designed for the exchange of position data typically resulting from in-field survey operations such as geophysical (including seismic), hydrographic etc.

A P1/11 file consists of two main sections, the header (containing the metadata and geodetic identity for the survey) and the data records.

1. To export Node information in P1/11 seismic data format, Click the button **P1/11** on the Node Dashboard tab. This will open the P1/11 Export Configuration page.

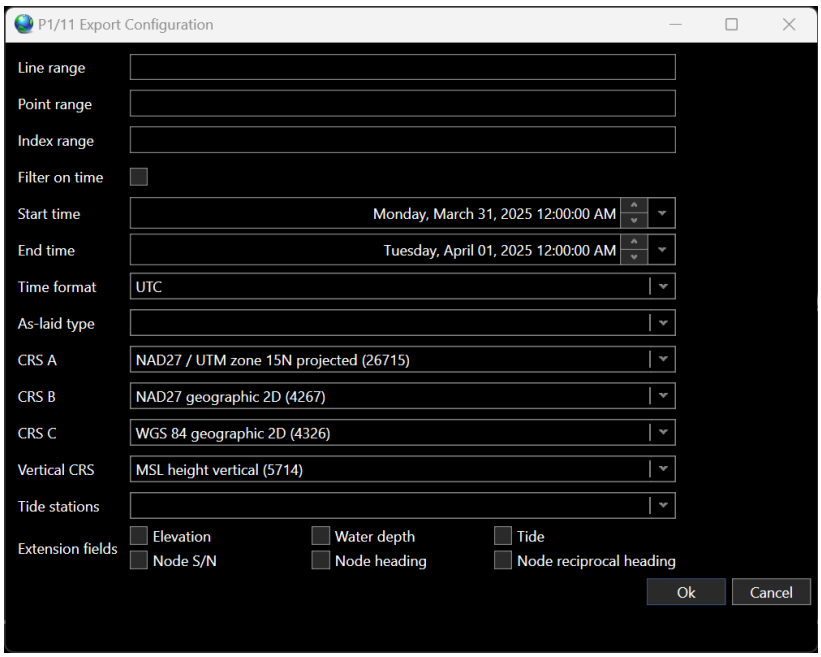


FIGURE 115 P1/11 EXPORT CONFIGURATION PAGE

- **Line range:** Enter single line or line range. Line range entered as i.e. 1601-1633

- **Point range:** Enter single point or point range. Point range entered as i.e. 1609-1729
- **Filter on time:** A time window for data export. Set **Start time** and **End time**
- **Time format:** Select timestamp for export. Options available are Local, UTC, GPS or Julian
- **As-laid type:** Select node state to be exported. Options include Deployed, Retrieved or Both
- **CRS A:** Select projected CRS (**Required**) to be included in export
- **CRS B:** Select geographic CRS (**Required**) to be included in export
- **CRS C:** Select (Optional) geographic CRS to be included in export
- **Vertical CRS:** Select vertical CRS to be included in export
- **Tide stations:** Select tide station that has been added in Database Services (Tides)
- **Extension fields:** Select addition items to be included in export

2. Click Okay to save the (*.p111) file.

10.4 EXPORT TEMPLATES

Node Dashboard export templates allow for highly efficient and fully customized data export. To access the template editing window, select the **Edit Template** button from the Node Dashboard ribbon tab.

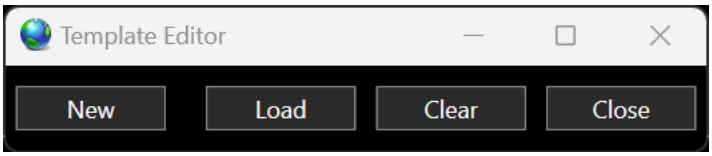


FIGURE 116 TEMPLATE EDITOR

- **New:** Create a new template and enter the template editor window with this template
- **Load:** Load an existing template from file into the editing window
- **Clear:** Clear the existing template from the window, discarding any changes
- **Close:** Close the template editor

10.4.1 TEMPLATE EDITOR VIEW

The template editor view consists of 4 tabs: Fields, File Location, File Schema and Custom Header. Export button to save the template as a json file and Import button to import template configuration.

10.4.2 FIELDS TAB

On the fields tab, a list of available items is shown on the left hand side (Available Items). The Design, Deployed, Recovered, and Other sections may be expanded to access items below each one. To add an item to the list of selected items, double click on the item in the available items list or select it and click the Add button.

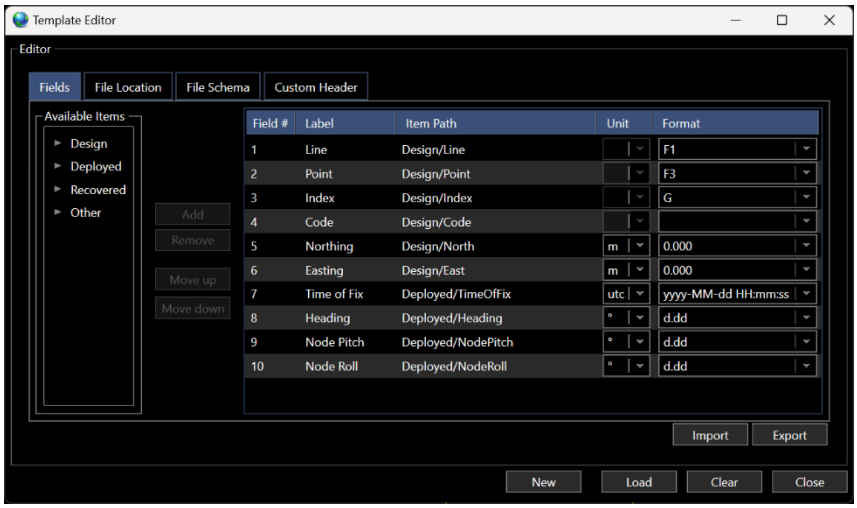


FIGURE 117 TEMPLATE EDITOR WINDOW – FIELDS TAB

The Label field is editable and is optionally included in the output. If a field has been added which has unit options, such as distance having the unit of meter or feet, these options will appear in the Unit field.

The format column allows the user to select number of decimals or other formatting options for the output display.

Note: Refer to https://learn.microsoft.com/en-us/powerquery-m/standard-numeric-format-strings for more information on numeric F, E, G, D and N formatting.

10.4.3 FILE LOCATION TAB

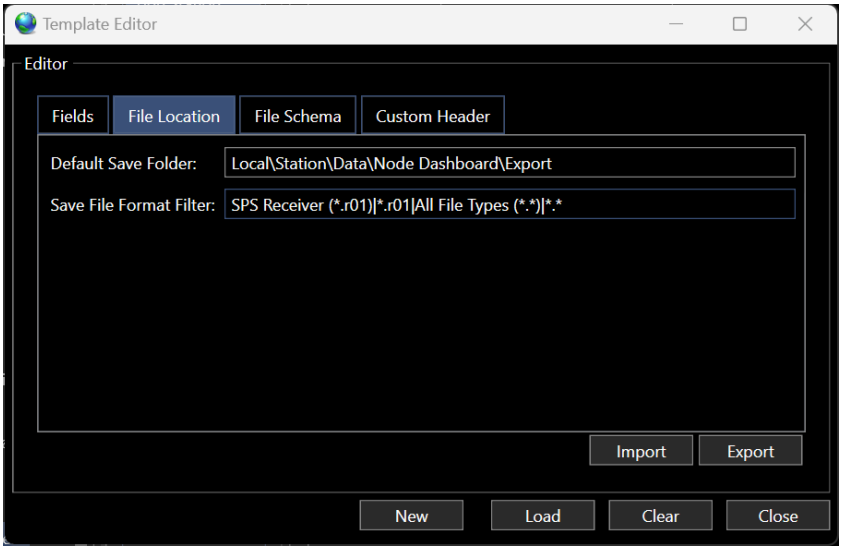


FIGURE 118 TEMPLATE EDITOR WINDOW – FILE LOCATION TAB

- **Default Save Folder:** The default save folder can be specified relative to the local project folder
- **Save File Format Filter:** The Save File Format Filter is used when saving the output files. This is a customizable string which is parsed to create options in the Save File dialog for the output data, see Figure 120.

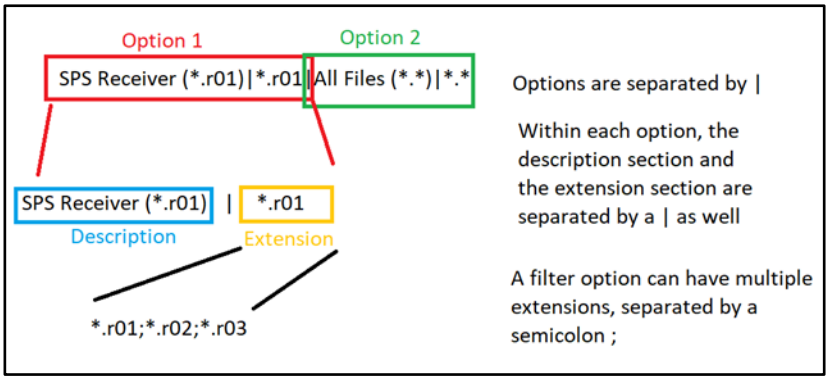


FIGURE 119 FILE FORMAT FILTER SETUP

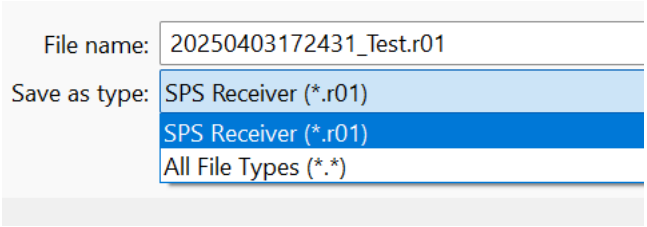


FIGURE 120 SAVE FILE FILTER RESULT EXAMPLE

10.4.4 FILE SCHEMA TAB

On the File Schema tab, the user can select how the fields will be arranged in the output file Delimited or Fixed Width.

10.4.4.1 FILE TYPE – DELIMITED

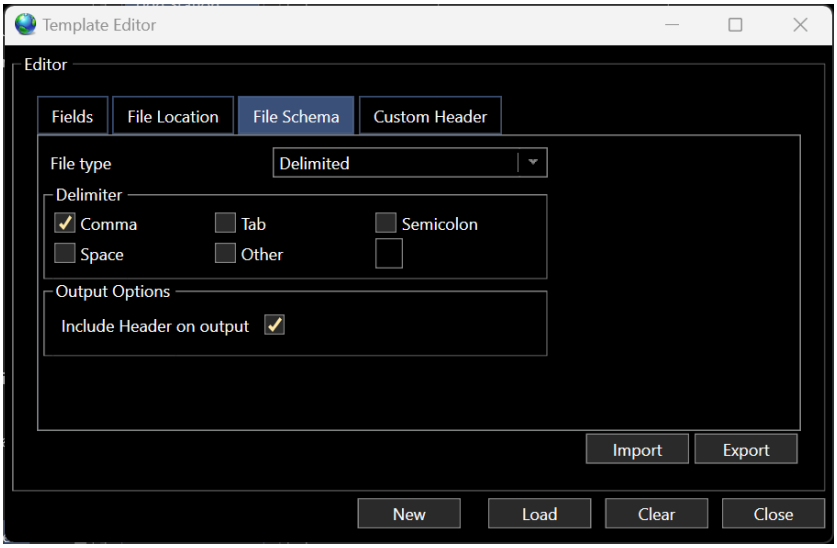


FIGURE 121 FILE SCHEMA – DELIMITED

- **Delimiter:** Specify the Delimiter
- **Output Options:** If this option is selected, the labels from the Fields tab will be output as the first row in the file, after any custom header

10.4.4.2 FILE TYPE – FIXED WIDTH

For the fixed width setup, see Figure 122, the field # on the first column corresponds to the Field # in the Fields tab, see Figure 117.

Click the Width for each field to change how many cells the field will occupy in the output file.

Click in the Alignment column to switch between the options of Left, Center, and Right.

To move a specific field up and down the list, change its order on the Fields tab. The Up/Down setup on the File Schema tab refers to moving up and down the field alignment definitions, which refer to the configuration of width and alignment.

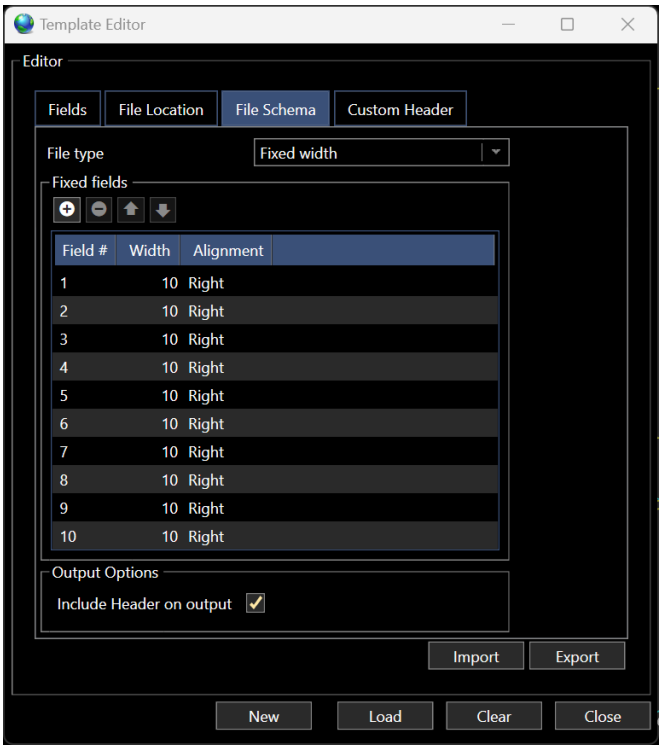


FIGURE 122 FILE SCHEMA – FIXED WIDTH

10.4.5 CUSTOM HEADER TAB

The Custom Header tab allows the user to insert text which will end up in the exported file. In the example screenshot below, a standard SPS header has been pasted in to allow for creation of an SPS file, see Figure 123.

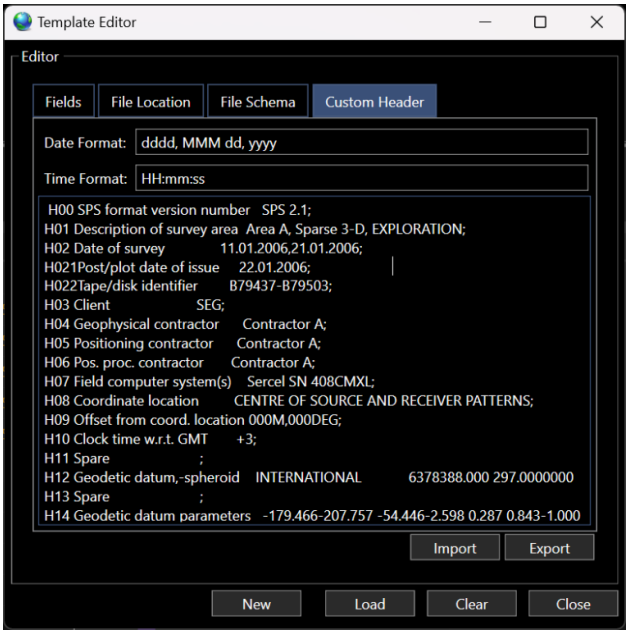


FIGURE 123 CUSTOM HEADER TAB

Certain substitutions may also be made, to replace parts of the preset header with data. These substitutions are made by including special key words within curly braces {}.

The following substitutions are available:

- **{DATE_UTC}** replaced with the current date in UTC, using the format specified in the Date format box at the top of the view
- **{DATE_LOCAL}** replaced with the current local date, using the format specified in the Date format box at the top of the view
- **{TIME_UTC}** – replaced with the current time in UTC, using the format specified in the Time format box at the top of the view
- **{TIME_LOCAL}** – replaced with the current time local time (with the zone defined in windows clock settings)
- **{FILENAME}** – the filename chosen during export
- **{CRS_TYPE}** – type of CRS (projected, geographic, geocentric)
- **{GEODATUM}** – description of the geodetic datum, for example: SIRGAS 2000[4674]SIRGAS 2000 6378137.000 298.2572221

10.5 EXPORT

With templates set up, it is now possible to export data using those templates. The export option is found on the Node Dashboard tab of the ribbon in the Export Templates section, Export button. This opens the Detailed Summary Report setup.

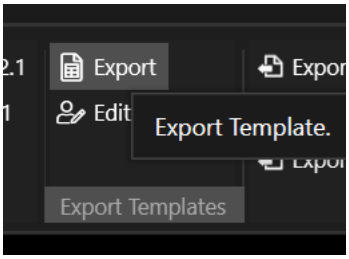


FIGURE 124 EXPORT USING TEMPLATE

1. The first page of the Detailed Summary Report setup is the option for Data Filtering (Time and Data).

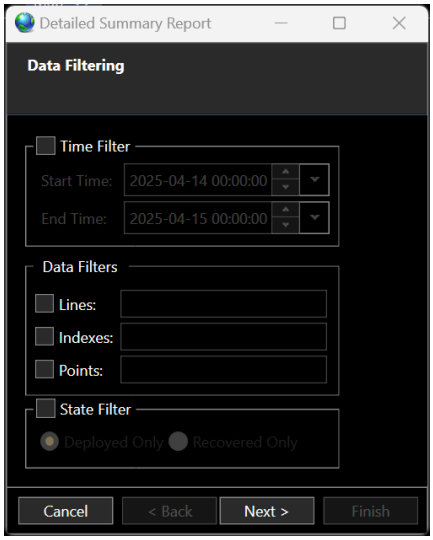


FIGURE 125 EXPORT – DATA FILTERING PAGE

2. Click Next, this opens the Tide Correction page. The user can optionally apply tide to elevations in the export. Here, a tide station and interpolation method can be selected, see Figure 126.

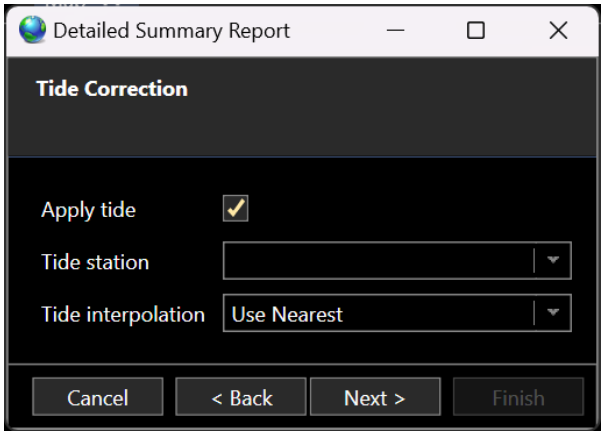


FIGURE 126 EXPORT – TIDE CORRECTION PAGE

3. Click Next to open the Template Selection page. The Template Selection page allows the user to select one or multiple export templates. This allows more than one export file to be created at the same time – such as a CSV file and SPS file.

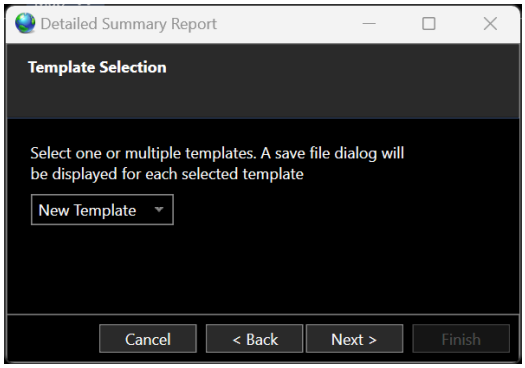


FIGURE 127 EXPORT – TEMPLATE SELECTION PAGE

4. Click Next to open the Position Units page, see Figure 128. The Output CRS of the data is chosen for any coordinate outputs; the units are determined buy the Output CRS selection. The default is the Working CRS.

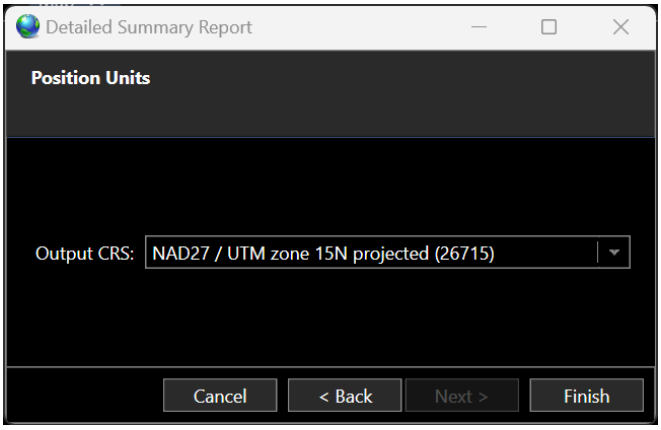


FIGURE 128 EXPORT – POSITION UNITS PAGE

5. Click Finish, a save file dialog for each template selected in the wizard will be presented, allowing for quickly customizing the file names.

11 DATA EXCHANGE

Data Exchange enables the user to transfer files between Node Dashboard systems. .

11.1 EXPORT NODES

1. Click on the Export Nodes button in the Data Exchange section.

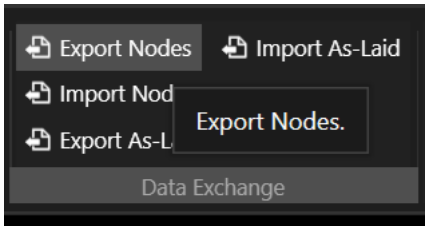


FIGURE 129 DATA EXCHANGE – EXPORT NODES FILE

2. Enter file name to save the Node Transfer File (.4nt)

11.2 IMPORT NODES

1. Click on the Import Nodes button in the Data Exchange section.

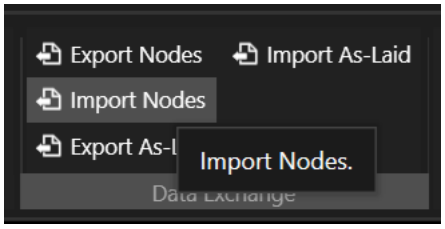


FIGURE 130 DATA EXCHANGE – IMPORT NODES FILE

2. Select Nodes file to import from the open file page.

11.3 EXPORT AS-LAID

1. Click on the Export As-laid button in the Data Exchange section.

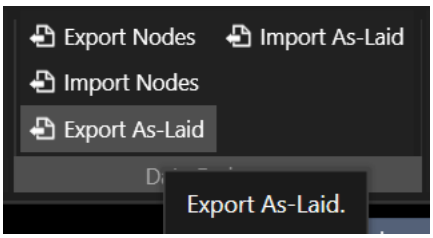


FIGURE 131 DATA EXCHANGE – EXPORT AS-LAID FILE

2. Save the file as a Multiseis Json format or a As-laid Transfer file (.4at).

Note: The Multiseis export is the same as the operation export shown in 3.3.3. When exporting a 4at (4D nav As-Laid Transfer) format file, the time period for the export may be specified. It is recommended when collaborating between vessels to do regular export/imports of small files at a time.

11.4 IMPORT AS-LAID

1. Click on the Import As-Laid button in the Data Exchange section.

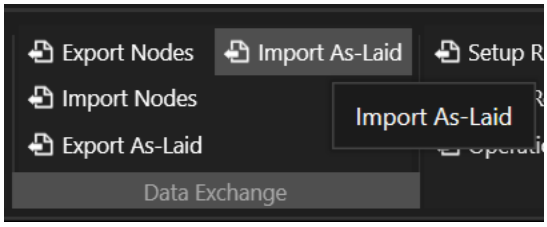


FIGURE 132 DATA EXCHANGE – IMPORT AS-LAID FILE

2. Select As-laid file to import from the open file page.

The *.4at format is a JSON based proprietary NavView format for sharing Node Dashboard As-Laid information between NavView systems.

Note: To use this feature, the same version of NavView Node Dashboard must be in use on both sides of the exchange. A *.4at file will also include preplot details such that if an as-laid fix was taken on a preplot then exported, and imported into another system which did not have that preplot, the preplot would be created.

When importing *.4at files, as-laids included in the file that are not already present on the importing system, will be populated. For example, if the importing system has a deployed as-laid for Preplot 1001-1001, and the 4at file has a different copy of that fix, nothing will be

imported for that item. However, if the file had a recovery fix on the same preplot, it would be imported to the system.

12 REMOTE OPERATIONS

Remote operations allow a remote station to trigger node deployments/recoveries.

The primary NavView, which has the ROV configuration, has to act as the server, and the other NavViews connect to it via the GRPC configuration, so it is outside of the regular Network Services.

From the Setup tab on the ribbon there is a section for GRPC Remote Control, configure Server and configure Client option.

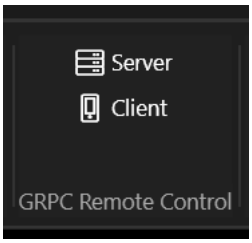


FIGURE 133 GRPC REMOTE CONTROL – NAVVIEW SETUP TAB

On the main NavView, configure as the server.

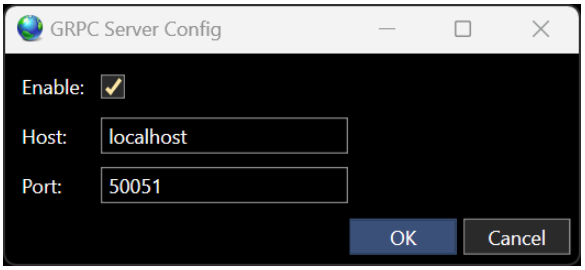


FIGURE 134 GRPC SERVER CONFIGURATION

On the remotes, configure as a client to point to the IP address of the main NavView PC.

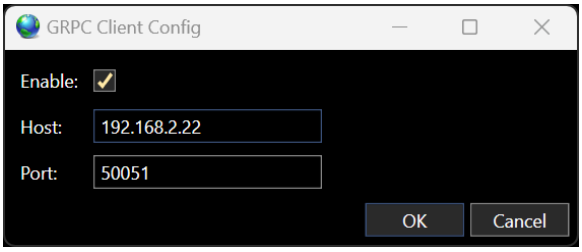


FIGURE 135 GRPC CLIENT CONFIGURATION

Remote ROV window may be opened for each ROV from the Node Dashboard ribbon, on the client NavView station.

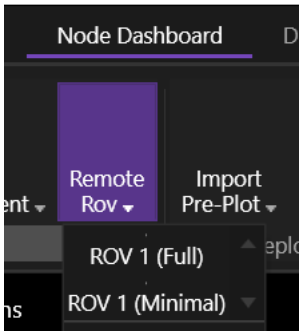


FIGURE 136 REMOTE ROV SELECTION

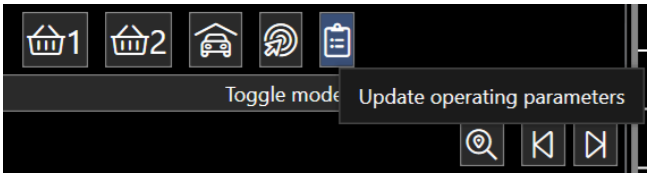
There is the (Full) management as well as the (Minimal) management option.

The Remote ROV window is similar to the regular ROV Management window. There is a connection logo at the top left of the view  which will turn red and indicate any error messages if a connection cannot be established.

The (Full) is intended for having a dedicated surveyor fix taking station - maybe two online surveyors simultaneously on different stations for example.

While the (Minimal) is for the ROV pilots who don't set up the line and tray themselves but have someone do it for them.

Note: The Operating Parameters configuration is only available using the (Full) management option.



Note: When taking a fix from the remote window, only the snapshot fix model is available. Furthermore, no fix confirmation view appears on screen – the fix is accepted automatically. The online surveyor on the main station may monitor progress by keeping the As-Laid Live history window open and watching for fix updates as they come in.