SITEWORKS

SITE CONTROLLER SOFTWARE

USER GUIDE

Version 1.00
Revision A
April 2018

Trimble

Corporate Office

Trimble Inc.
935 Stewart Drive
Sunnyvale, CA 94085
USA
www.trimble.com

Heavy Civil Construction Division

Trimble Civil Engineering & Construction
10368 Westmoor Drive
Westminster, Colorado 80021
USA

800-361-1249 (Toll Free)
+1-937-245-5154 Phone
+1-937-233-9441 Fax
www.trimble.com

Email: trimble_support@trimble.com

Legal Notices

Release Notice

This is the April 2018 release (Revision A) of the Trimble Siteworks Software User Guide. It applies to version 1.00 of the software.

Product Limited Warranty Information

For applicable product Limited Warranty information, please refer to the Limited Warranty Card included with this Trimble product, or consult your local Trimble authorized dealer.

Safety Information

Before you use your Trimble product, make sure that you have read and understood all safety requirements.

⚠ WARNING – This alert warns of a potential hazard which, if not avoided, could result in severe injury or even death.

⚠ CAUTION – This alert warns of a potential hazard or unsafe practice that could result in minor injury or property damage or irretrievable data loss.

NOTE – An absence of specific alerts does not mean that there are no safety risks involved.

Vehicle safety

⚠ WARNING – When you select the Vehicle mode, the following warning message appears:

WARNING: Do not operate Siteworks while driving the vehicle. Failure to heed this warning may result in a collision causing property damage or personal injury.

Do not interact with the touch screen, keyboard, or software in any way while the vehicle is moving. While the vehicle is moving, the software provides a continuous display of position and data that can be seen at a glance. Operating the device or interacting with the software while the vehicle is moving can be a distraction for the operator, and may result in collision causing property damage or personal injury.

Contents

1

Introduction

This manual provides you with the necessary information to perform measurement and stakeout tasks with the Trimble® Siteworks site controller software. To perform the many tasks that the Siteworks software is capable of, you need the Siteworks site controller software running on a Trimble controller that is either connected to a Precision or Location GNSS system, or to a total station. You can also use the internal GPS of the controller to locate objects. The Siteworks software is the field software that runs on a Trimble TSC7 controller, Trimble T10 Tablet or the Trimble Site Tablet 10. For presentation and training purposes, a software emulator that runs on a Microsoft® Windows® 10 Pro operating system is also available on the Siteworks webpage of the Trimble website (www.construction.trimble.com/siteworks).

Even if you have used other GNSS, GPS and total station products before, Trimble recommends that you spend some time reading this manual to learn about the special features of this product. If you are not familiar with GNSS, GPS, or total stations visit the Trimble website (www.trimble.com).

Overview

The Siteworks software is a site measurement tool that streamlines earthworks and surface finishing operations. It enables construction contractors to measure material volumes, monitor grades and laid material thicknesses, and to perform various site layout tasks such as point, line, and surface stakeout, in addition to survey measurement of points, lines, and surfaces.

Start up and set up of the sophisticated GNSS and total station technology is quick and easy. The software manages data for multiple project sites, single large project sites, and large sites that have been divided into zones. When a field engineer, grade checker, or surveyor opens a work order, the software opens all the data files needed to complete that work order. Because the software delivers results immediately, informed decisions can be made in the field.

The Siteworks software can also be used to check site grading operations that have been performed using a machine control system, such as a Trimble GCS900 grade control system or Trimble Earthworks. If your organization does not have a 3D machine control system, the system provides site control, grade checking, progress volumes, and stakeout capabilities to facilitate earthmoving operations.

The field engineer can quantify progress, check data, and set out the information needed to keep the machines moving. The Siteworks system tracks activities at each site and keeps a continuous record of all results. Related data is stored, together, as a permanent construction record and is output as a TXT or DXF file. Back at the office, the software delivers comprehensive operation analysis data.

Related information

Sources of related information include the following:

- Release notes – The release notes describe new features of the product, information not included in the manuals, and any changes to the manuals. They can be downloaded from Technical Information / Documentation section on the Trimble website.
- Trimble training courses – Consider a training course to help you use your GNSS system to its fullest potential. For more information, go to the Trimble website at www.trimble.com/Support/Index_Training.aspx.

Technical support

If you have a problem and cannot find the information you need in the product documentation, contact your local dealer. Alternatively, go to the Support area of the Trimble website (www.trimble.com/Support/). Select the product you need information on. Product updates, documentation, and information relating to support issues are available for download.

2

Starting the Software

Installing the software

The software and its modules are downloaded and installed through the Trimble Installation Manager (TIM), which can be downloaded from www.trimble.com/installationmanager/. For the TSC7, T10 Tablet and SiteTablet 10 devices, TIM is installed directly on the devices, and the devices must be connected to the Internet via Wi-Fi or Ethernet through the available docking stations, or USB to Ethernet adapters.

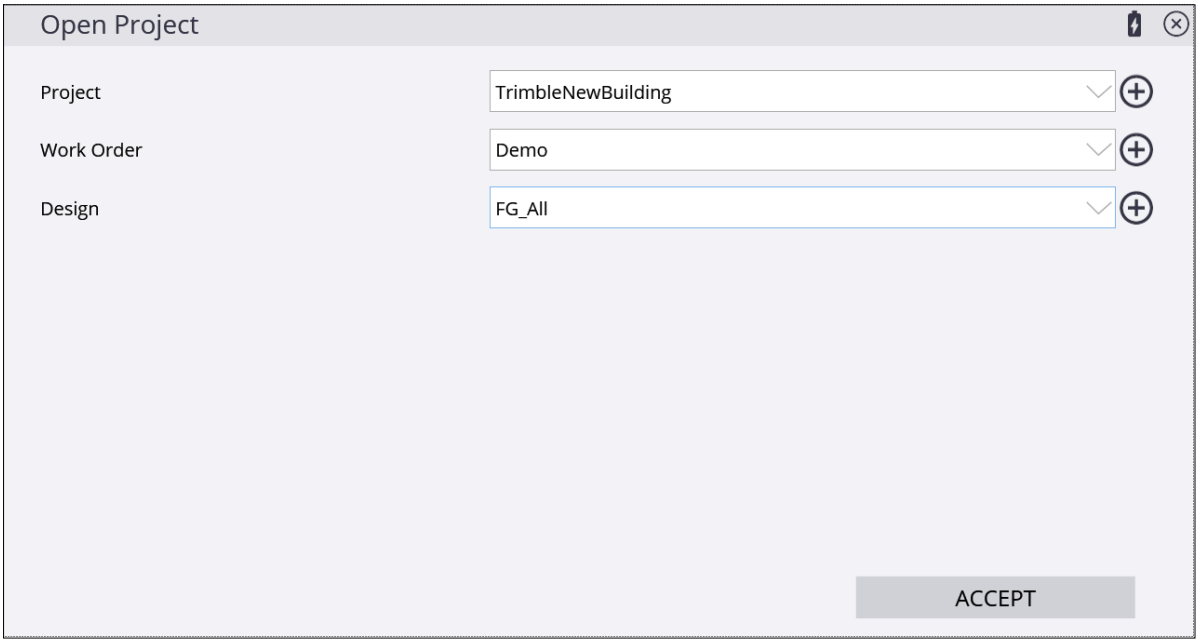
TIM needs to be installed directly on the device, and the device must be connected to the Internet. The License Manager checks which software options were purchased for the specific controller with this serial number and checks for the latest version of the software that is available for this controller.

Note that the Siteworks version that the controller is entitled to is determined by the serial number of the controller and the status of the software warranty expiration date. The latest versions of the Siteworks software are only available to users with software warranty expiration dates within one month of the release date of the software. Please contact your dealer if your controller is out of warranty. For brand new controllers, the warranty period begins when the Siteworks software is first downloaded to the device through a successful connection to the Trimble Installation Manager.

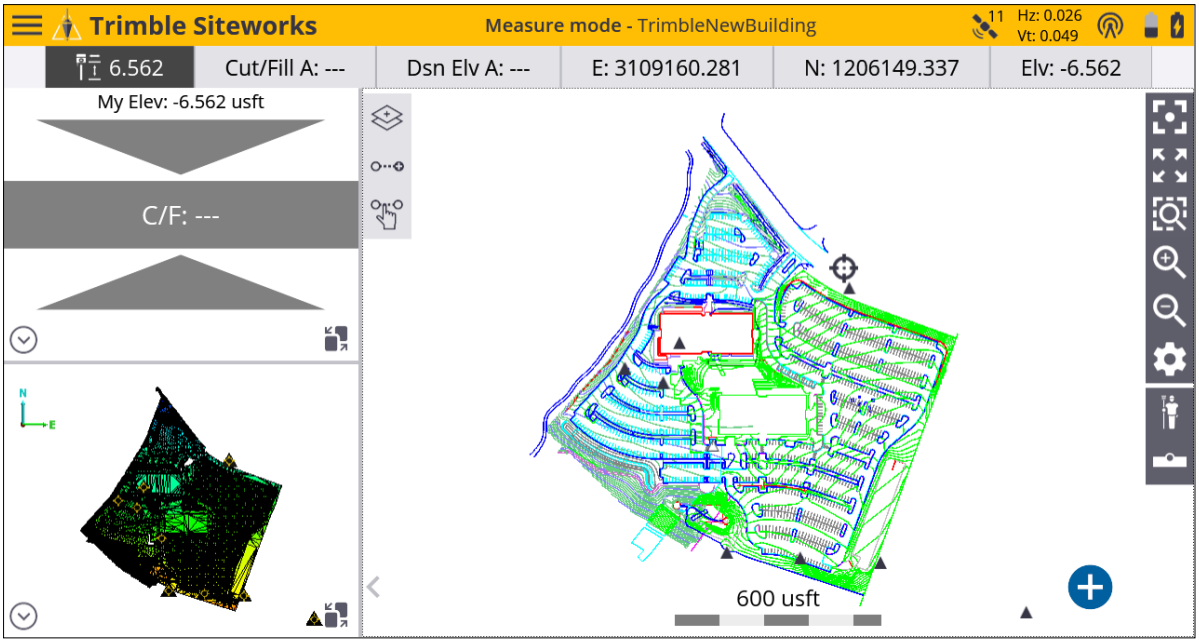
Starting the software

To start the Siteworks software on your controller, double-click on the Siteworks icon on the controller's desktop. The Siteworks software will start in the Open Project dialog where you can select an existing project, design, and work order on your controller or create a new one.

The last used Project, Work Order and Design will be displayed. Click on the down arrow to the right of the fields to select a different Project, Work Order or Design, or click on the plus symbol to create a new one.



After accepting the selection, the software loads your data and the map view. If you have previously connected a rover receiver to the site base station or were using a total station, the software tries to automatically connect with the last used device configuration.



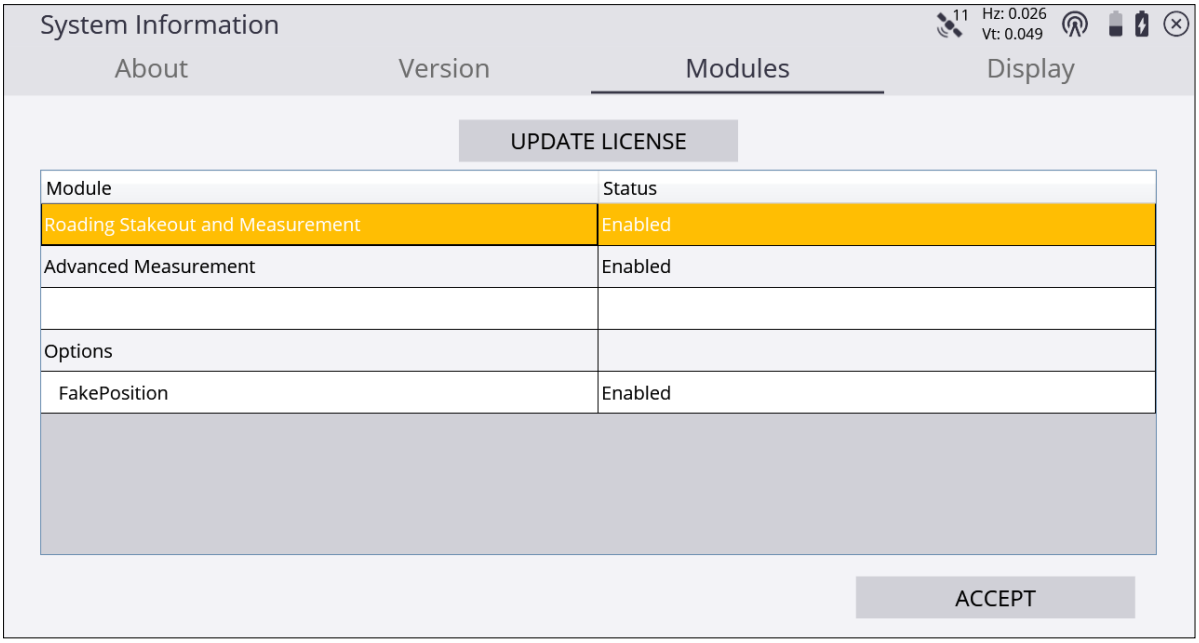
If the software is not automatically connecting to your positioning device, tap on the **Project Setup** menu by going to the **main menu** and tap **Connect Device**and select either **GPS** or **Total Station** . The **Receiver Setup** menu opens to start the connection to the positioning device.

System information

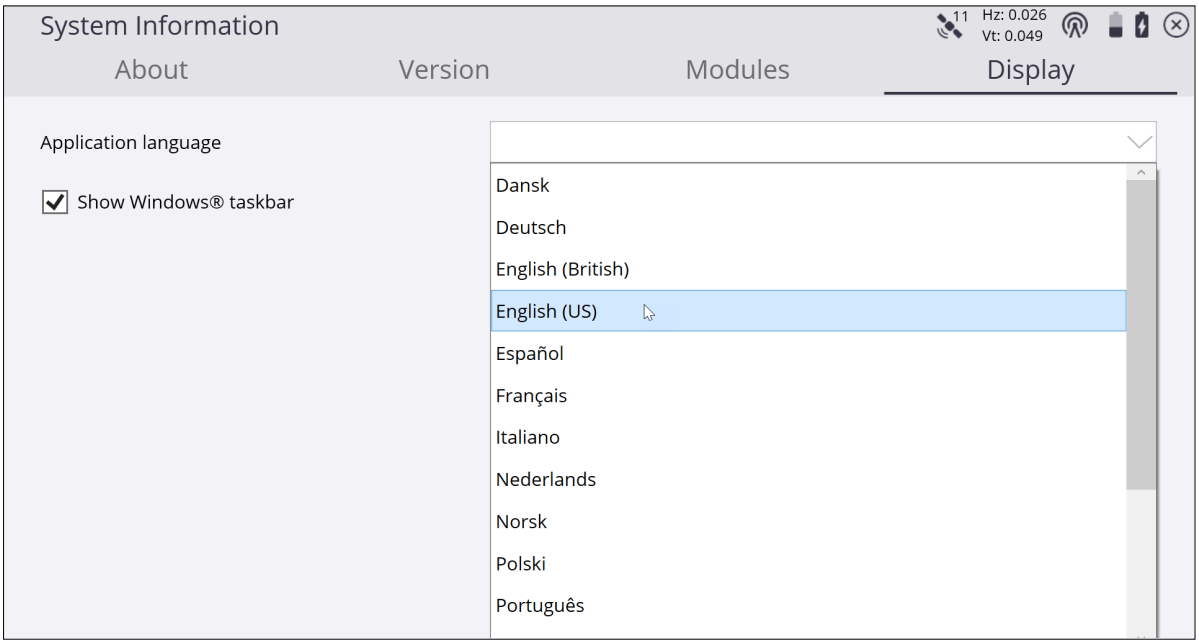
The **System Information** dialog has multiple tabs that contain information about the current version of the Siteworks software, which modules are enabled, which positioning sensors are connected, which firmware the sensors have, and what language to display.

From the main menu, select **Settings** / **System Info**. You must have an open site and work order to access this menu.

If you have purchased a module after the controller was activated, activate the new module by tapping **Update License** in the **Modules** tab.



In the **Display** tab, you can toggle between the different supported and installed languages:



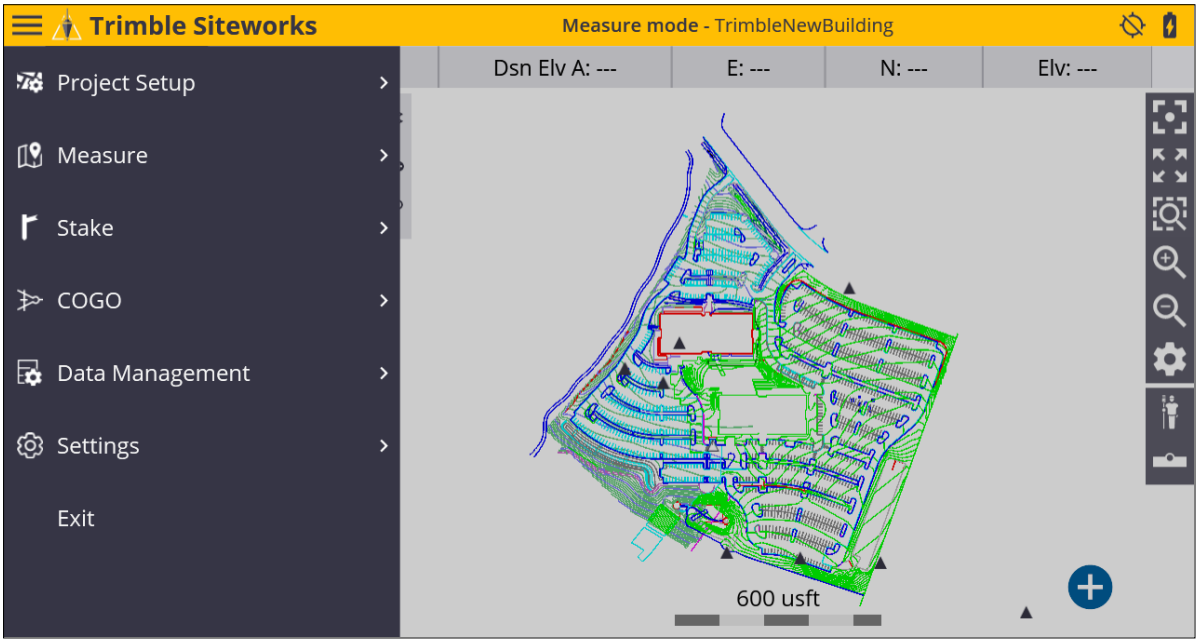
3

Menus

The Siteworks software is a menu-driven system. From the **Map** screen you can access the main menu through the **Home** button  at the top left of the screen (*also known as the Hamburger menu*).

Home menu

The **Home** menu is the main menu of the Siteworks software. It contains the following buttons:



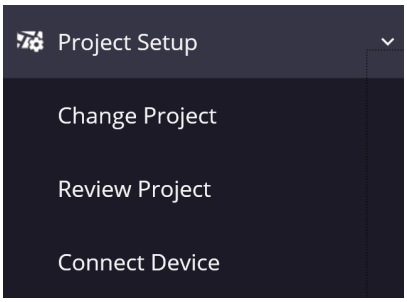
To access it from any map screen, tap ☰.

The Project Setup menu

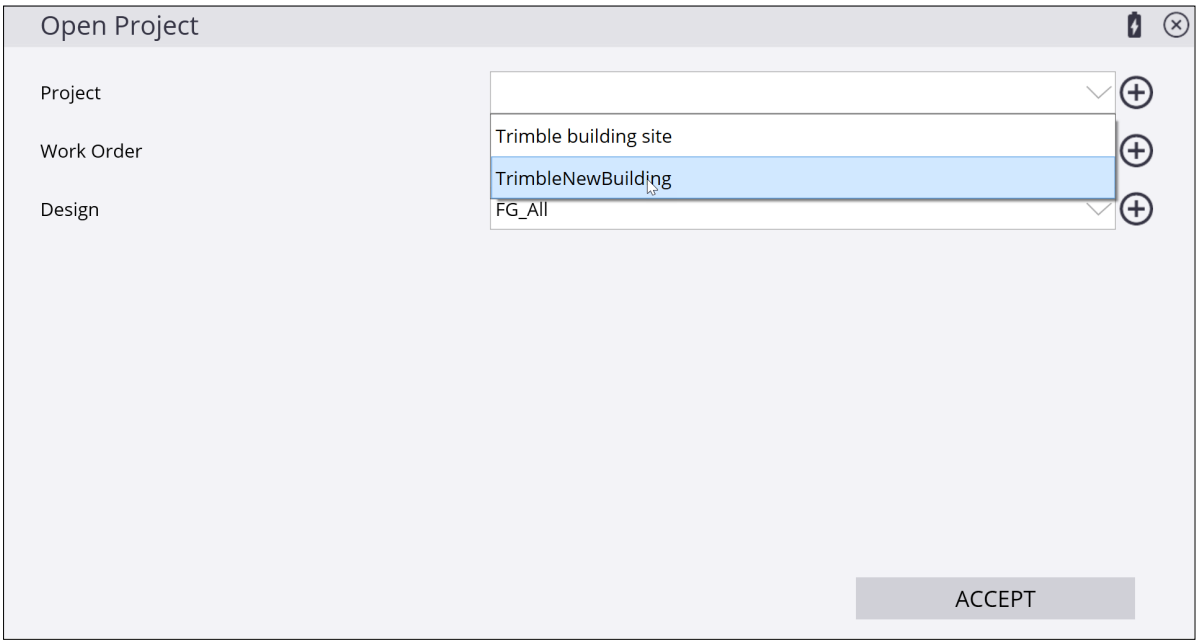
The **Project Setup** menu contains options to manage the details of the project, the tools to create, open, review, and select projects, designs, and work orders. You can also change the design referenced by the current work order and create new designs.

The change and review project buttons are used to create, open, review, and select sites, designs, and work orders. You can also change the design referenced by the current work order and create new designs.

From the **Home** menu, tap **Project Setup**. This shows the options **Change Project** and **Review Project**:

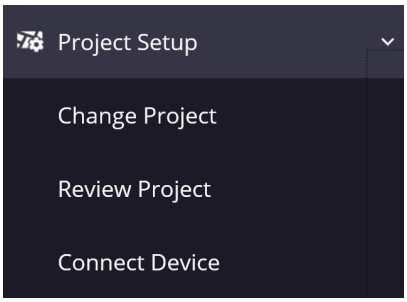


Tap **Change Project** to create a new project, design, and work order, or select from existing sites on the controller. At the top of each list, the software displays an option to create a new project, design, or work order, the last three items that have been used, and then all remaining data on the controller:

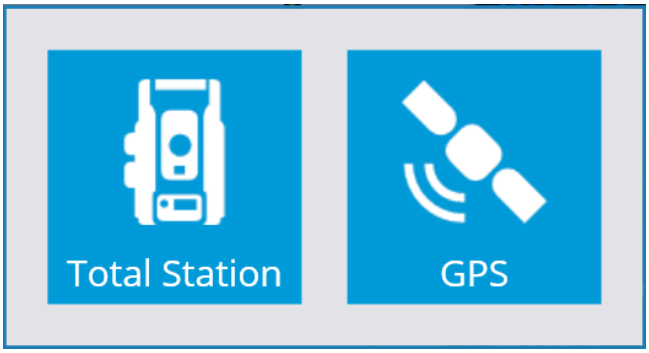


Tap **Review Project** to show the names of the currently loaded project, work order and design, along with the **Settings** of distance and angle units, coordinate order and other site settings.

The third menu **Connect Device** is used to connect a total station or GPS to Siteworks.



Select **Connect Device** and then either **Total Station** or **GPS**.



This takes you through the setup of the selected instrument. For information on the use of these, see Measuring with GPS, page 127 or Measuring with a Total Station, page 154.

When a device is connected, the tabs under **Project Setup** change. These different menus are outlined below. See The GPS menu, page 19 and The Total Station menu, page 20.

The GPS menu

When a GPS is connected to Siteworks, use this menu to connect to the rover receiver, carry out a site calibration, and select a predefined coordinate system. You can also recheck the system setup on a control point.

From the **Home** menu, tap **Project Setup**.

The following commands are available:

Command	Description
Connect Device	Set up and start a GNSS base station, rover receiver, or start the internal GPS of the controller.
Coordinate System	Select a predefined coordinate system from the coordinate system library.
Project Calibration	Perform or resume a single-point, two-point, or multi-point site calibration, and review the calibration when it is completed.
Recheck System	Check the existing site calibration and base station location by checking in on a known control point.

The Total Station menu

When a total station is connected to Siteworks, use this menu to connect to the instrument and carry out total station setup to establish the position and orientation of the instrument. General functions in this menu include being able to recheck the system setup on a control point or set the total station into Machine Control mode.

From the **Home** menu, tap **Project Setup**.

The following commands are available from this menu:

Command	Description
Connect Device	Connect to the total station using either a cable, Bluetooth® wireless technology, or a 2.4 GHz radio for robotic operation. After a connection is established, this button changes to a **disconnect** button, that when pressed will disconnect from the total station and place it into standby mode.
Total Station Setup	Set up the total station and establish its position and orientation on the site using a known control point or arbitrary location (also known as free station or resection) methods.
Machine Control Setup	Set up instrument for machine control.
Recheck System	Check an existing instrument setup on a known control point.

The Measure menu

The software always initially starts in Measurement mode. Use this option to switch between stakeout and measure modes.

From the **Home** menu, tap **Measure**.

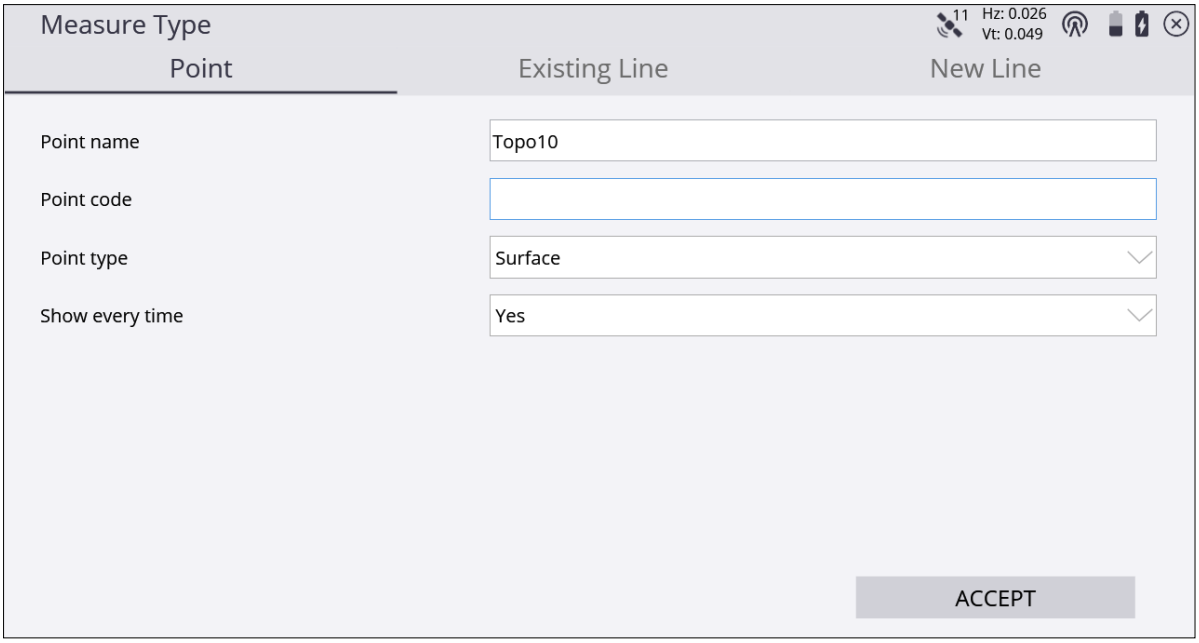
In this menu, you can accomplish all site measurement functions including:

- Grade checking
- Material thickness checking
- Topographic measurements to create surface models (for example, volume computations)
- Site point and line feature measurements to record the location of non-surface features
- Real-time cut/fill information against a selected design model
- Stockpile Scanning if a total station is connected

Before you use this menu, you must connect and set up your GNSS or total station in the **Home** menu. If you have not yet done a system setup when you select an option in this menu, the software automatically prompts you to go through the station establishment process for either a total station or the rover setup for GNSS.

In the **Point type** of the **Measure Type** tab, you can define if measured points are to be stored as part of a Digital Terrain Model (DTM) surface or only as a site feature and not included in any surfaces. This menu can also be accessed from the main screen by clicking the measure type button

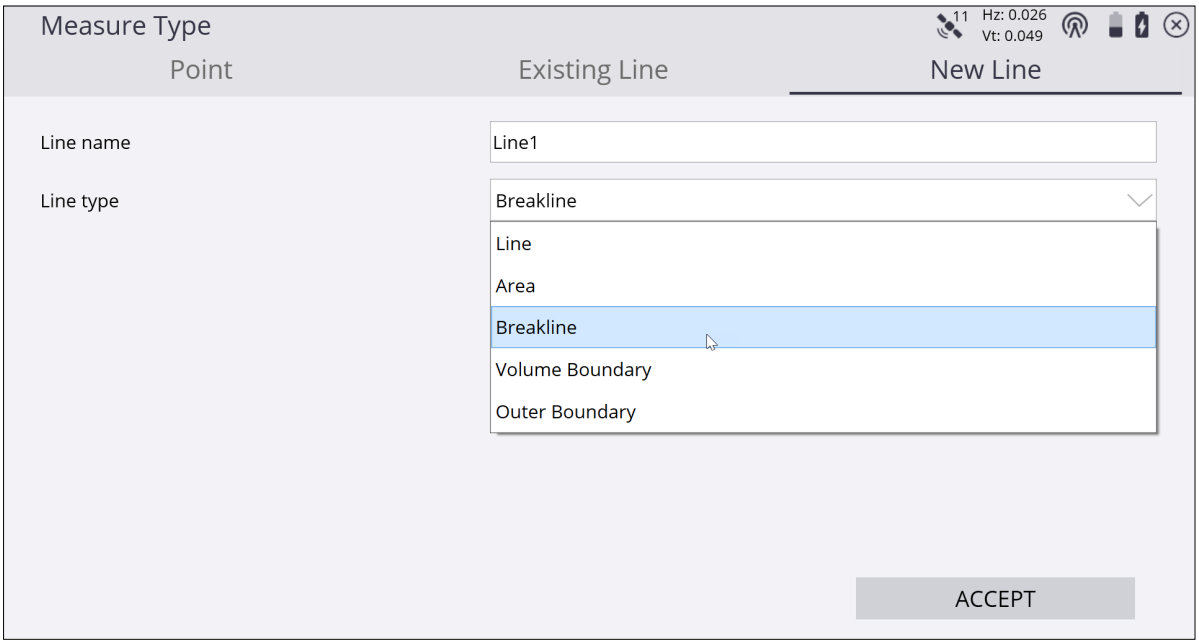
Select **Surface** or **Feature** from the **Point type** field to choose if the point or line will be included in a surface or just stored as a feature. When the Advanced Measurement module is installed, the measurement type is controlled by the settings in the FXL file Feature Code Library for each feature code.



Lines are measured by selecting one of the Line tabs: **New Line** or **Existing Line**.

Selecting **Existing Line** enables the selection of an existing line from a table and the continuation of measuring that line from the last measured end point along it.

Selecting **New Line** requires entering a **Line Name**, and choosing what kind of line type it is. Line types that are included in the DTM surface measurement are Breaklines, Volume Boundaries, and Outer Boundaries. Lines and areas are not included in the DTM surface measurement.



The Stake menu

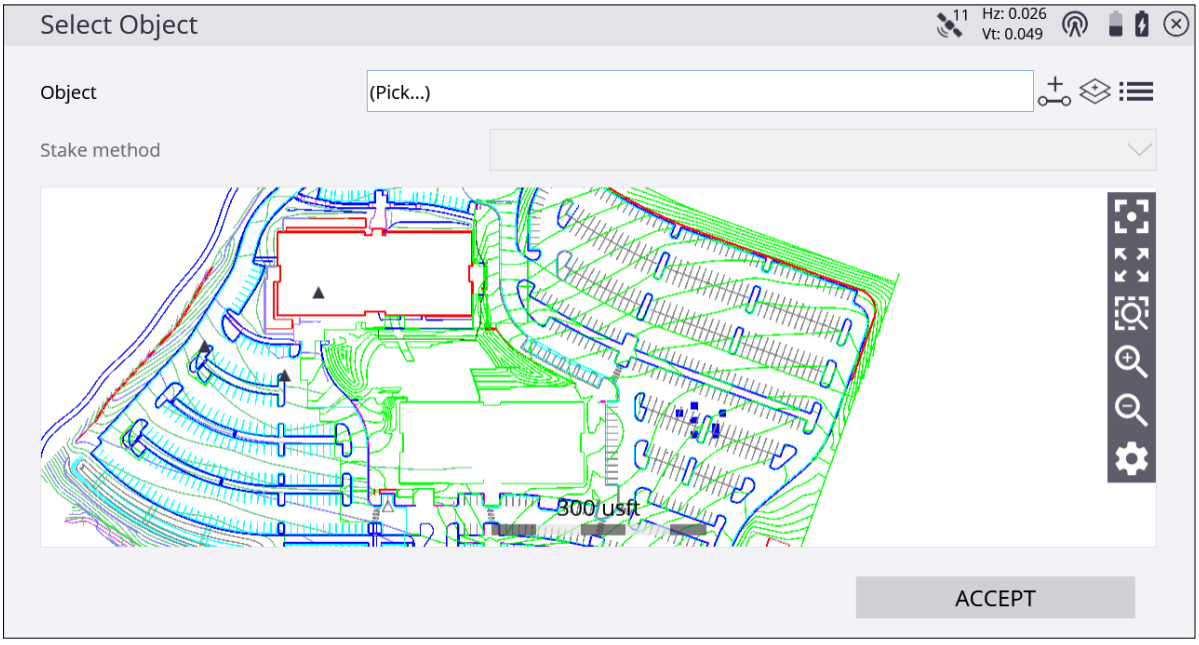
The **Stake** menu contains all stakeout functions including staking points, lines, alignments, surfaces, side slopes, catch points, and road features.

From the **Home** menu, tap **Stake**.

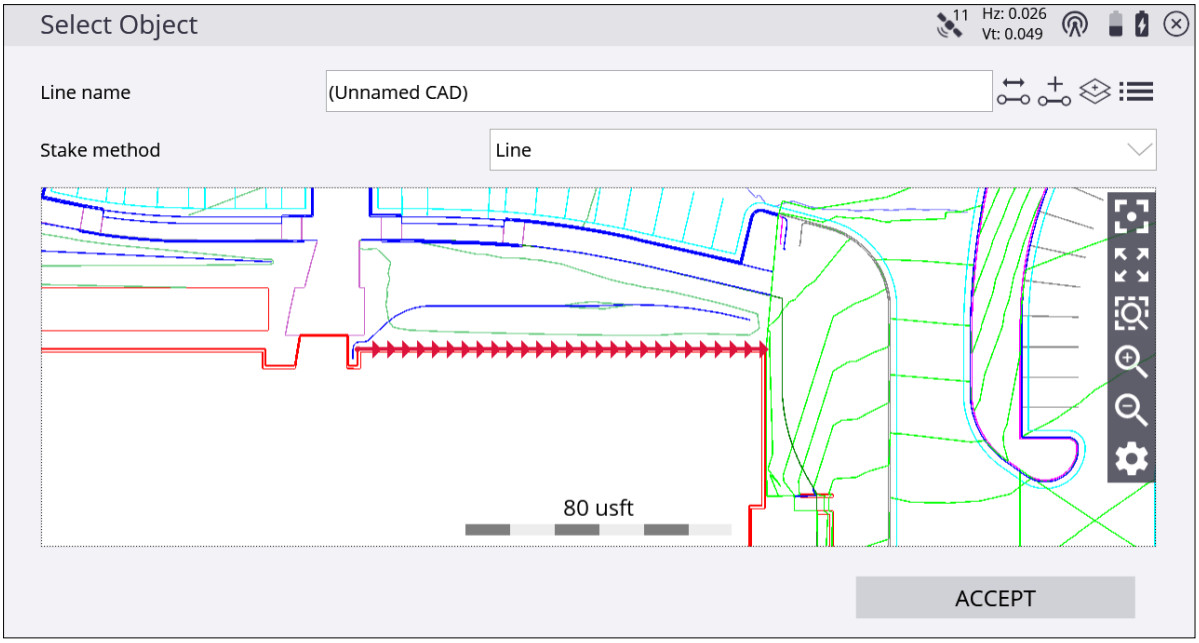
After entering the stakeout function, the **object selection** screen appears. Select the object that you want to stake out either by tapping directly on it in the map or from the list in the top right corner of the screen, by selecting .

To stake a user-defined plane, select .

To define a new line to stake, select .



After you select an object, multiple stake methods are available that differ depending on the type of object that you selected. For more information, see Stakeout Workflow, page 98.


Select Object
11
Hz: 0.026
Vt: 0.049
Line name
(Unnamed CAD)
Stake method
Line
80 usft
ACCEPT

The COGO menu

The **COGO** (Coordinate Geometry) menu contains a number of features, such as area, distance, bearing, slope, and point generation functions that can be used to generate points for stakeout operation from CAD or measured data in the currently loaded design and work order. The menu also provides access to review and edit functions for editing breaklines and deleting points or lines to resolve surface modeling problems. A point manager function is also available to view, edit, create, and delete points in a work order in a list format.

From the **Home** menu, tap **COGO**.

The following commands are available from this menu:

Command	Description
Review & Edit Data	View or delete point and lines, add breaklines and boundaries, compute distance, areas and angles, calculate surface volumes (stockpiles) or periodic progress volumes, generate contours of measured data, and enter 3D surface viewer mode.
Create Points/Arcs	Create points, lines, boundaries, arcs, and circles by a variety of methods including free point, bearing and distance, radius, and offset to a line. To create free points, tap the screen or enter its coordinates.
Key In Roads	Enter a basic road alignment and road cross sections in the field. This function requires the Roading module.

The Data Management menu

Use this menu to view and edit points, import and export data or selected design information to an external memory device such as a USB flash drive. An alternative is to use the Connected Community for real time data sharing. The menu also includes export functions for data use in the GCS900, Earthworks or AccuGrade grade control systems.

From the **Home** menu, tap **Data Management**.

The following commands are available from this menu:

Command	Description
Point Manager	View, create, edit, and delete control, stakeout, and design points in a list format of points in the currently selected work order.
Export Measured Data	Export measured data as a CSV custom point file format, DXF file, Record.txt file, or Network measurements for Advanced Measurement Module users.
Surface as Design	Write the surface data that was measured in the field to a new design surface as a .TTM file.
Export to Machine	Export a design to a CompactFlash card or USB drive for use with the GCS900 Grade Control System or Trimble Earthworks.Export measured data as a CSV custom point file format, DXF file, Record.txt file, or Network measurements for Advanced Measurement Module users.
Log	View a data log of the project to date including tasks performed and their detailed measurement settings. Import a comma, tab, or semicolon delimited point file in various formats. For more information, see page 53.
Grade Report	View a summary of all the grades checked. Export a design to a CompactFlash card or USB drive for use with the GCS900 Grade Control System.
Connected Community	Synchronize data with the Connected Community service using the Wireless Data Sync function.

Command	Description
Community Settings	Enter the device credentials to access the Trimble Connected Community service.

The Settings menu

Use this menu to change or view the current settings in the project. This is the menu where you can view the system settings and access multiple project settings such display, eBubble, and the various types of measurement settings.

From the **Home** menu, tap **Settings**.

The following commands are available from this menu:

Command	Description
System Info	Access the current software version, modules and license information and change the display language.
Map Display Settings	Change display options for measure mode, the design and the map layers. Change the screen view direction and switch which panels are displayed.
eBubble	Calibrate the eBubble and change the tolerance and sensitivity settings.
Measure Mode	Switch measurement modes between standing, walking, vehicle, static and EZ level.
Info Bar/Panel	Change what information is displayed in the information bar and panel.
Measure	View and change the measurement tolerances and settings.
Stakeout	View and change the stakeout tolerances and settings.
Roads	View and change the settings when in a project utilizing road stakeout.
Surface Settings	View and change the settings for a surface.
Second Surface	View and change the settings when a second surface is being used.

Command	Description
Reference Line	Select and set the settings for a reference line.
Avoidance Zone	Set the tolerances for the avoidance zones loaded into the project.
Catch Point Marking	View and change the settings to be used when staking a catch point.

The Exit menu

Exit

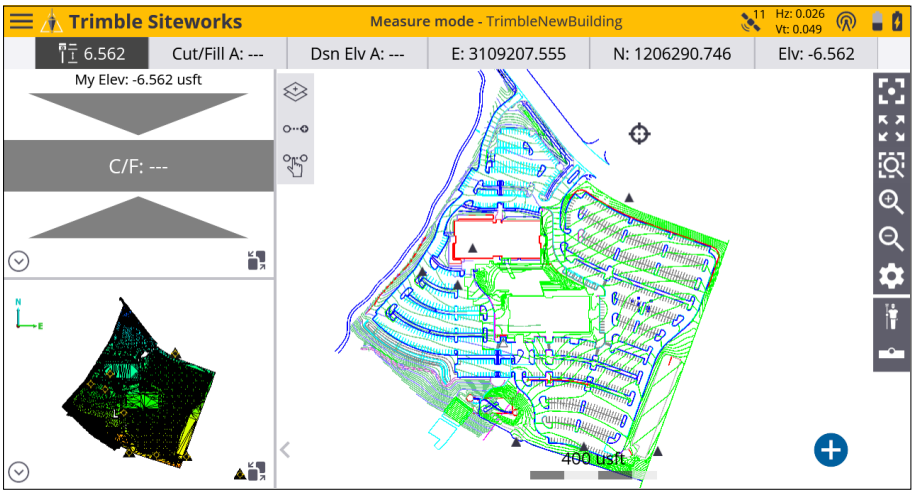
This menu closes the Siteworks software, and optionally can power down the receiver.

From the **Home** menu, tap **Exit**.

4

Measurement Screen

The Siteworks software is a menu-driven system. From the **Map** screen you can access the main menu through the **Home** button at the top left of the screen. The map screen contains three panels. One large panel called the main panel, and two small panels on the left side of the screen called the top panel and bottom panel. These can be set up to display different types of information such as a plan view, 3D surface viewer, eBubble and cross sections.



Measure modes

The Measure mode controls a number of functions for the GPS receiver and total station. There are multiple modes available, which you can switch between by tapping the Measure Mode icon on the right side of the main panel below the map control bar.

GPS measure mode

GPS measure mode	Name	Description
	Standing	Walk to a point and then take a single measurement.
	Walking	Walk the site and continuously take measurements. Points are measured based on a setting for horizontal distance and elevation change, or at regular time intervals.
	Vehicle	Drive the site in a vehicle. Points are stored based on a setting for horizontal distance and elevation or at regular time intervals. This feature contains an additional function to measure the antenna height by parking the vehicle over a known elevation point.
	Static	Measure a point for a longer period and to achieve a greater accuracy through position averaging.
	EZ Level	Graphically display elevations on a site relative to an arbitrary benchmark.

Total Station measure mode

Total Station measure mode	Name	Description
	Standing	Walk to a point and then take a single measurement.
	Walking	Walk the site and take continuous measurements. Points are stored based on a setting for horizontal distance and elevation change or at regular time intervals, which can be set by tapping the Measure mode icon either in the status bar or in the Trimble icon menu.
	Vehicle	Drive the site in a vehicle. Points are stored based on a setting for horizontal distance and elevation change or at regular time intervals, which can be set by tapping the Measure mode icon either in the status bar or in the Trimble icon menu. This feature contains an additional function to measure the antenna height by parking the vehicle at a known elevation point. Measurements are acquired at 3 Hz for all prisms except the MT1000 and MT900. These prisms are capable of producing measurements at 20 Hz.
	Averaging	Measure a point in multiple faces/rounds and display the standard deviation between measurements.
	DR	Measure a point using a DR reflectorless total station.

Total Station measure mode	Name	Description
	DR Target	Measure a point using a DR reflectorless total station that adds a user-defined distance amount, up to 5 m, along the straight line beam path from where the beam reflects off the surface.
	DR Scanning	Operate the total station in a scanner mode by manually turning the total station and setting a distance interval to measure points over.
	DR Averaging	Measure a point in DR mode using multiple faces/rounds and display the standard deviation between measurements.
	EZ Level	Graphically display elevations on a site relative to an arbitrary benchmark.

The icon below the **Measure mode** icon on the right side of the main panel below the map control is the **measure type** icon. This is used to select either point, existing line or new line.

When in stakeout mode, this icon changes to enable you to select the stake method. The options in this tab are discussed in Slope staking, page 108  .

Status bar

The status bar at the top right of the screen contains relevant information about the current status of the positioning system, battery status, measure mode, and measurement type. The icons change slightly depending on the positioning device. The following icons are typically available:

GPS mode

Icon	Description
	The radio connection status to the base station. Clicking on this provides access to the **Radio Information** menu. This icon changes depending on how corrections are being received.
	The number of satellites being tracked and the horizontal and vertical precision of the GNSS position solution. Clicking on this opens the **GPS status** menu.
	The battery level of the controller and externally connected GNSS receiver.

Total Station mode

Icon	Description
	Indicates the total station is in Target Tracking mode and provides access to the total station control panel. A lock button appears when the instrument is locked onto a prism.
	Indicates the total station is in Standard mode and provides access to the total station control panel. A lock button appears when the instrument is locked onto a prism.
	Shows the battery level of the controller and total station.

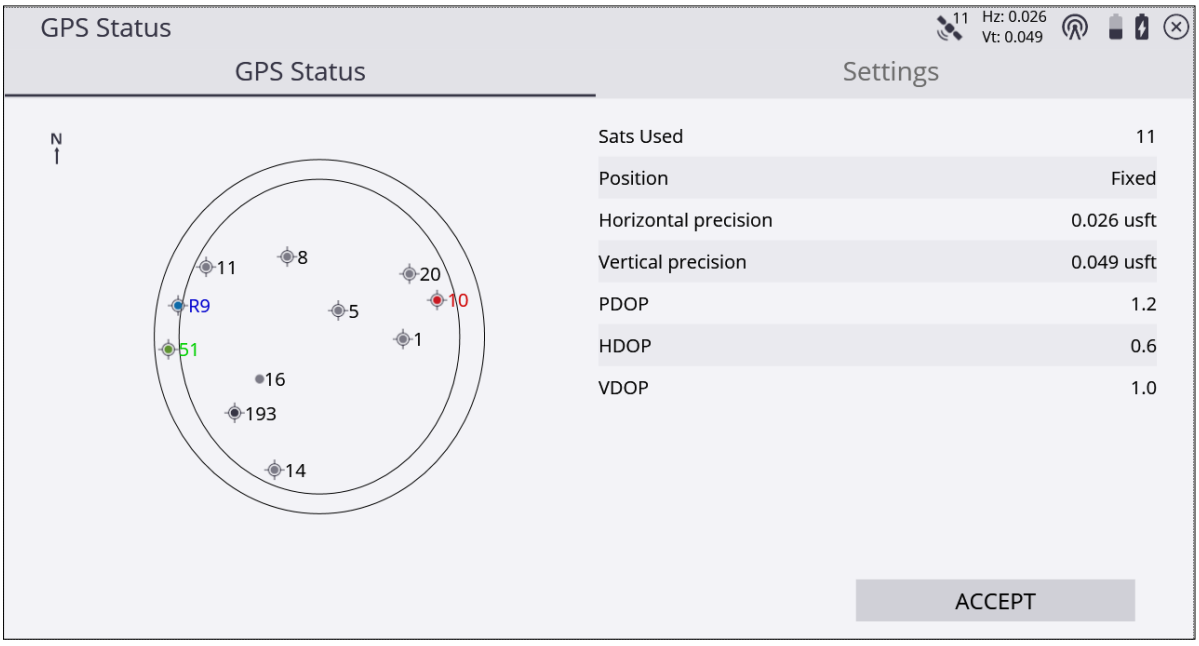
Precision GNSS control icons

When using the internal GPS or while connected to an external SPS receiver, an icon panel appears at the top right of the screen.

GPS mode

This icon...	shows...
	how many satellites are being tracked and the horizontal and vertical precision of the GNSS position solution.
	the radio connection status.
	the battery level of the controller and GNSS receiver.

Tap one of these icons to show more detailed information. To access the sky plot of the current satellite constellation, tap the Satellite or the Precisions icon:



The icons in the sky plot represent the following satellite constellations:

This icon...	shows...
	GPS

This icon...	shows...
	BeiDou
	Galileo
	GLONASS
	QZSS
	SBAS
	Satellite not used in position

The **Settings** tab, which appears when the controller is connected to an SPS receiver, is a shortcut to the **RTK Precision** option. If the precisions of the GNSS rise above these values, the info bars flash red and a warning pops-up if a measurement is attempted. The **GPS Only** check box enables you to toggle between using GPS and GNSS satellite constellations. The current Position, Horizontal, and Vertical Dilutions of Precision (DOP) values are also displayed.

While in the **GPS Status** screen, you can set the elevation mask of the GNSS rover receiver by pressing **Ctrl**+**M** on the controller's keyboard. Note that you must use the virtual Windows keyboard on a T10 or Site Tablet 10 to access the **Ctrl** key. The rover's elevation mask setting will also govern the elevation mask used for the satellites received from a base station, as the rover will ignore all satellite data from the base for those satellites below the mask elevation value set on the rover.

Tap the radio icon to open the **Radio Information** screen, where you can view information about the model of the radio, its current channel, the base name, and reception information. You can change the radio channel in this screen.

Map controls

The icons on the right of the main panel enable you to move around the screen and toggle information on and off to improve readability when there is a lot of information shown on the screen. They change depending on what data has been selected to be displayed in the main panel.

Plan View and Cross Section

Icon	Description
	Center the map on the current location.
	Zoom to the extents of the file.
	Zoom to a user-defined box (use a stylus to draw a box on the screen).
	Zoom in.
	Zoom out.
	View the map display options. Enables you to customize what is displayed on the map screen. You can choose the information that you want to view for the task rather than cluttering the screen with too much information.

Pinch-to-zoom capabilities are present across all panels if using a TSC7, T10 or Site Tablet 10 controller.

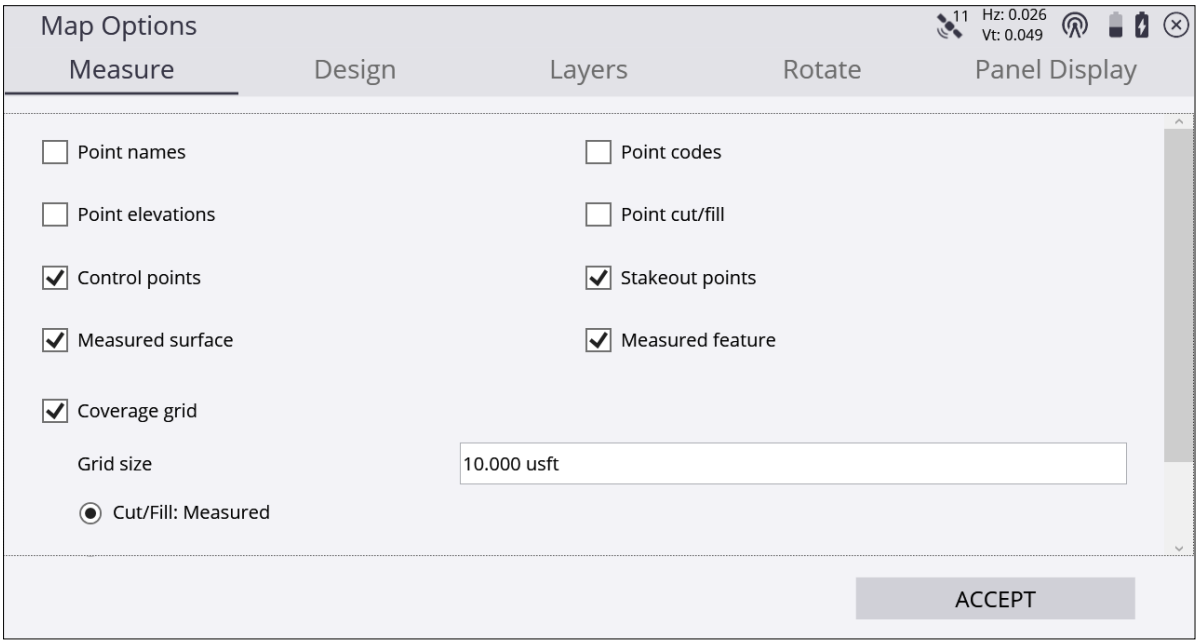
3D Surface Viewer

Icon	Description
	Center the map on the current location.
	Zoom to the extents of the file.

Icon	Description
	3d Orbit by clicking on the map and using the touch screen.
	Zoom. To zoom in, click this and swipe up on the touch screen. To zoom out, swipe down on the touch screen.
	Toggle on or off the TIN/ surface edges.
	Toggle on or off the stakeout flags.
	Open the **Surface Selection** menu to choose which surface to display.

Map Options: Measure tab

Use the **Measure** tab to filter the different measured data that is being displayed:

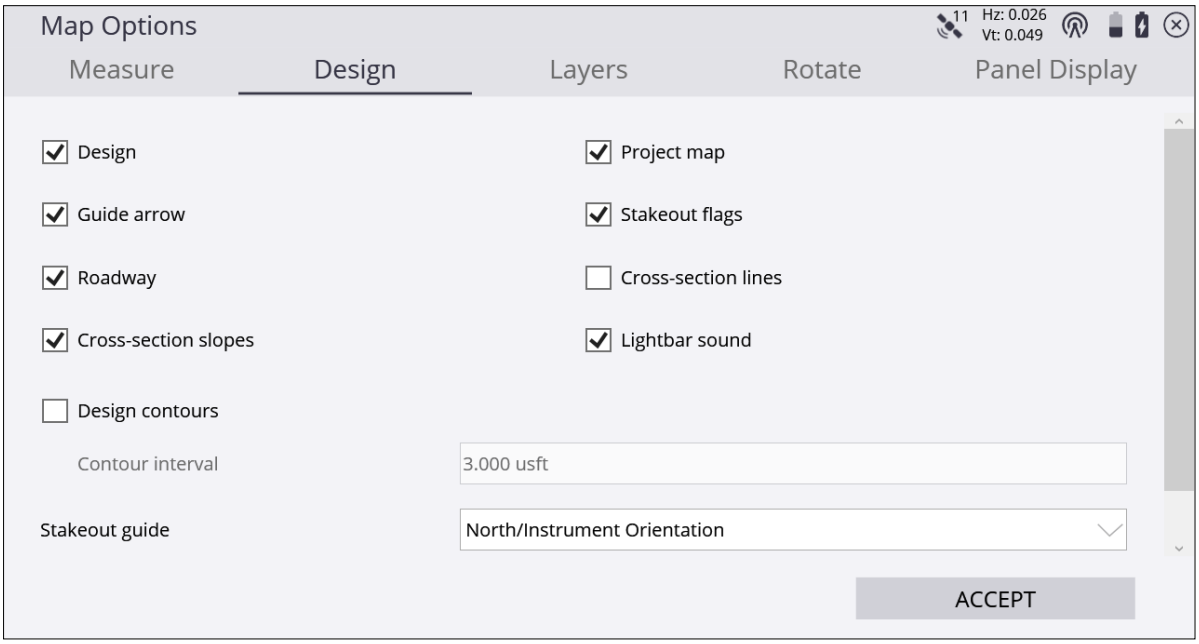


Select this option...	to display...
Point names	the point names of every point on the map view.
Point elevations	the point elevations of every point on the map view.
Control points	control points on the map view.
Measured surface	measurements that were recorded as a surface.
Coverage grid	a coverage grid map that shows cut/fill/in tolerance values as shades of red/blue/green respectively between a measured point and the selected design surface, between two design surfaces, or elevation changes as different shades of blue depending on the setting of the radio button. The Coverage Grid function has three options: • **Cut/Fill: Measured** – This setting displays a coverage grid based on the current measurement within a grid point relative to the design surface selected in the **Open Project** screen as different shades of red, blue or green. If there are multiple measured points within a grid cell, the largest cut/fill value is used in the color display.

Select this option...	to display...
	• Cut/Fill: Surface A-B – This setting displays a static coverage grid map of cut/fill between two design surfaces. Surface A is the primary design surface set in the Open Project screen. Surface B is the secondary surface set through the Home / Settings / Second Surface. This display is static and does not update based on any new measurements. It can only be displayed between two existing designs saved in the Site. • Elevation – This setting displays the relative elevation of grid cell points as shades of blue. Lighter shades of blue are lower in elevation than darker shades. The color scale gradations are distributed evenly across the difference between the maximum and minimum measured elevations. The colors automatically update if a new elevation value is measured. • Grid Size – Using this setting the size of the coverage grid can be changed and set to a scale to meet the needs on site.
Point codes	the point codes of every point on the map view.
Point cut/fill	the cut/fill information for every point on the map view.
Stakeout points	stakeout points on the map view.
Measured feature	measurements that were recorded as a site feature.

Map Options: Design tab

Use the **Design** tab to filter the different design data types that are displayed:

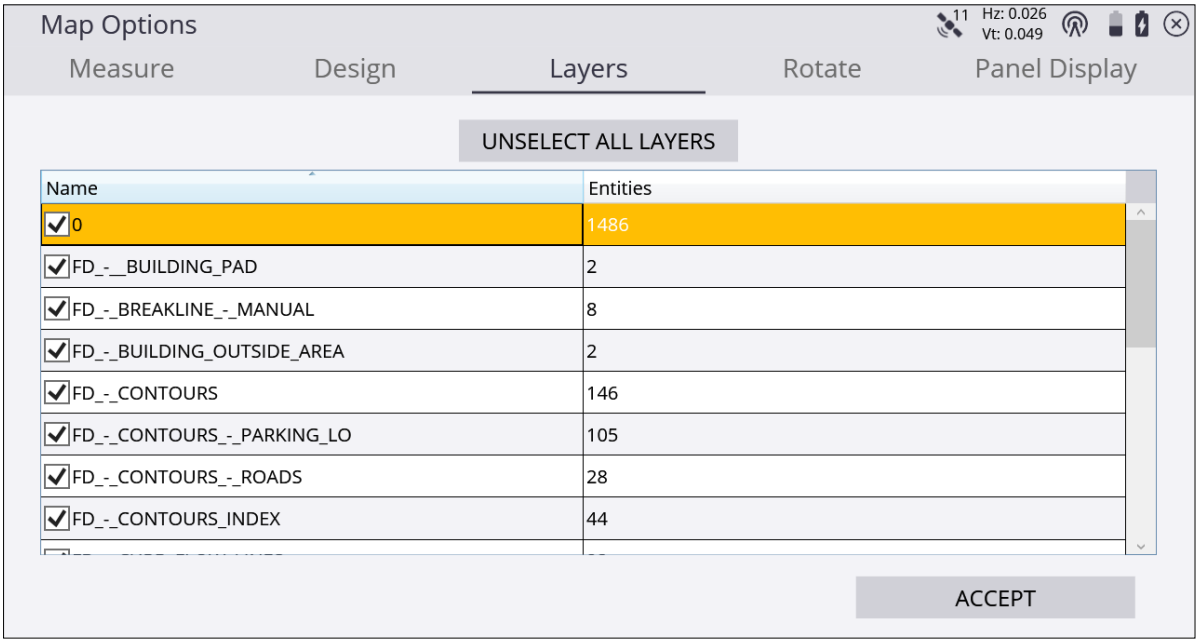


Select this option...	to display...
Design	the design map on the map view.
Guide arrow	the guidance arrow to navigate to points of interest.
Roadway	the center line of any roadway design.
Display cross-section	display the slope values for each segment of a road cross section.
Project map	the project map on the map view.
Stakeout flags	stakeout flags on the map view.
Cross section lines	the cross sections of any loaded roadway design.
Lightbar Sound	an indicator sound when approaching the design elevation.
Design contours	contours on the map view, if a design is loaded.
Contour interval	sets the contour interval for the displayed design contour lines.
Stakeout guide	sets the behavior of the fine stakeout guide bullseye when within 2 m of a stakeout point. **North/Instrument Orientation** will keep the stakeout guide oriented towards north for GNSS or as if facing the total station. **Approach** will orient the guide with “up” as

Select this option...	to display...
	in the direction of approach towards the point, when the guide appears at 2 m from the point. **Hide** will not show the guide.
Cut/Fill precision	determines the number of decimal places displayed in the info bars on the map screen and for the Cut/fill of individual points if the Point Cut/fill is selected in the **Measure** tab.

Map Options: Layers tab

Use the **Layers** tab to turn individual or all layers on and off in the design map to improve readability if there is a lot of data in the current design DXF file.

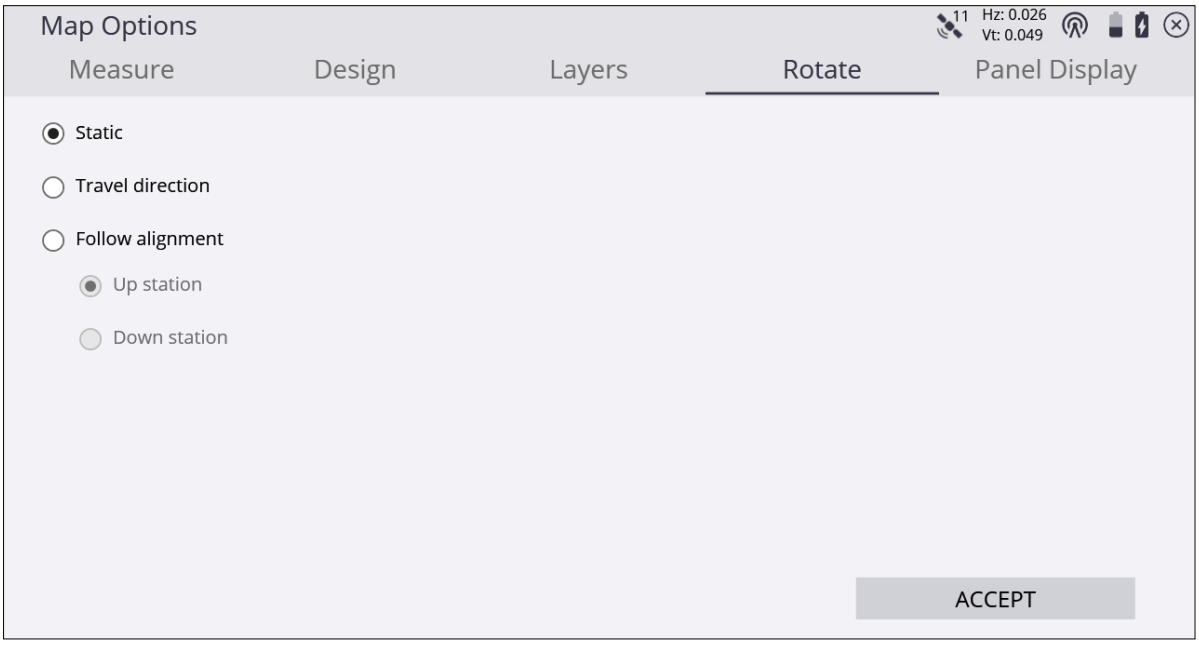


To toggle on and off individual layers, select/deselect the check box next to their names.

To toggle on and off all layers, click **Select/Unselect All Layers** at the top of the screen.

Map Options: Rotate tab

Use the **Rotate** tab to control the map rotation of the **Measurement** screen:

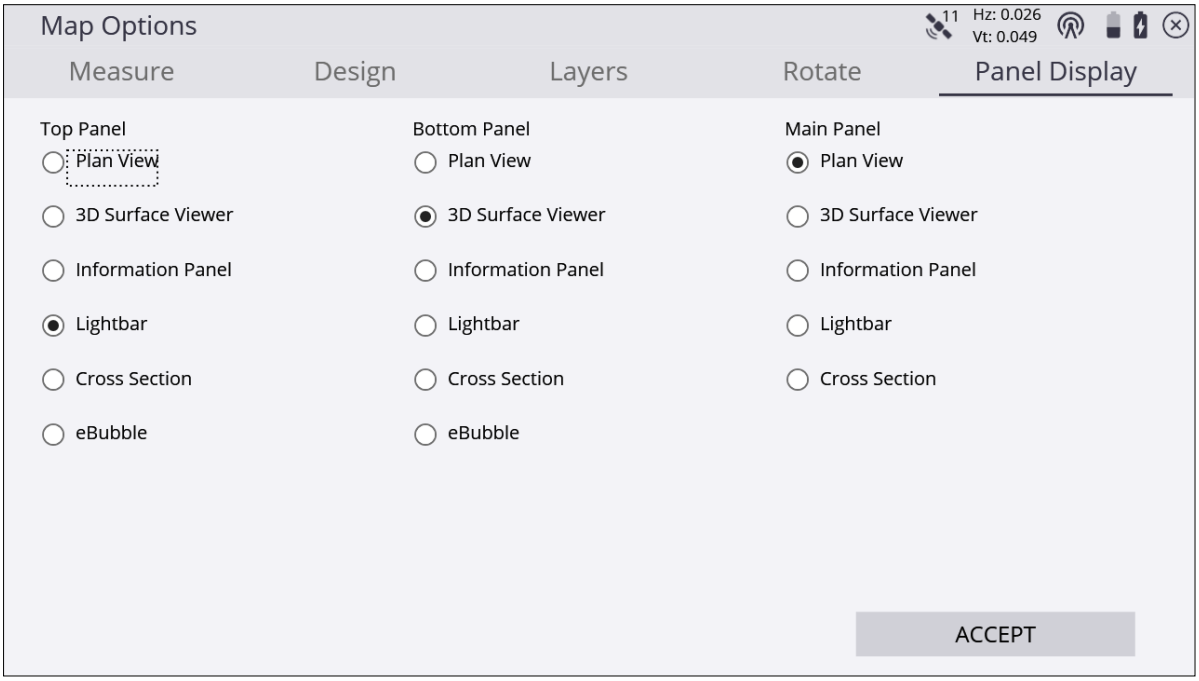


Select this option...	to orientate...
Static	the map view to North.
Travel direction	the map view to your direction of travel/walking.
Follow alignment	(This option is only available when an alignment is selected.)
Up station	the screen up station.
Down station	the screen down station.

NOTE – When any option other than Static is selected, a North arrow automatically appears on the screen.

Map Options: Panel Display tab

Use the **Panel Display** tab to control what information is shown in each of the panels on the main screen:



Select this option...	to orientate...
Plan View	Displays the plan view in the chosen panel
3D Surface Viewer	Displays the 3D surface viewer in the chosen panel
Information Panel	Displays the information panel in the chosen panel
Lightbar	Displays the lightbar in the chosen panel
Cross Section	Displays a selected cross section in the chosen panel.
eBubble	Displays the eBubble in the chosen panel

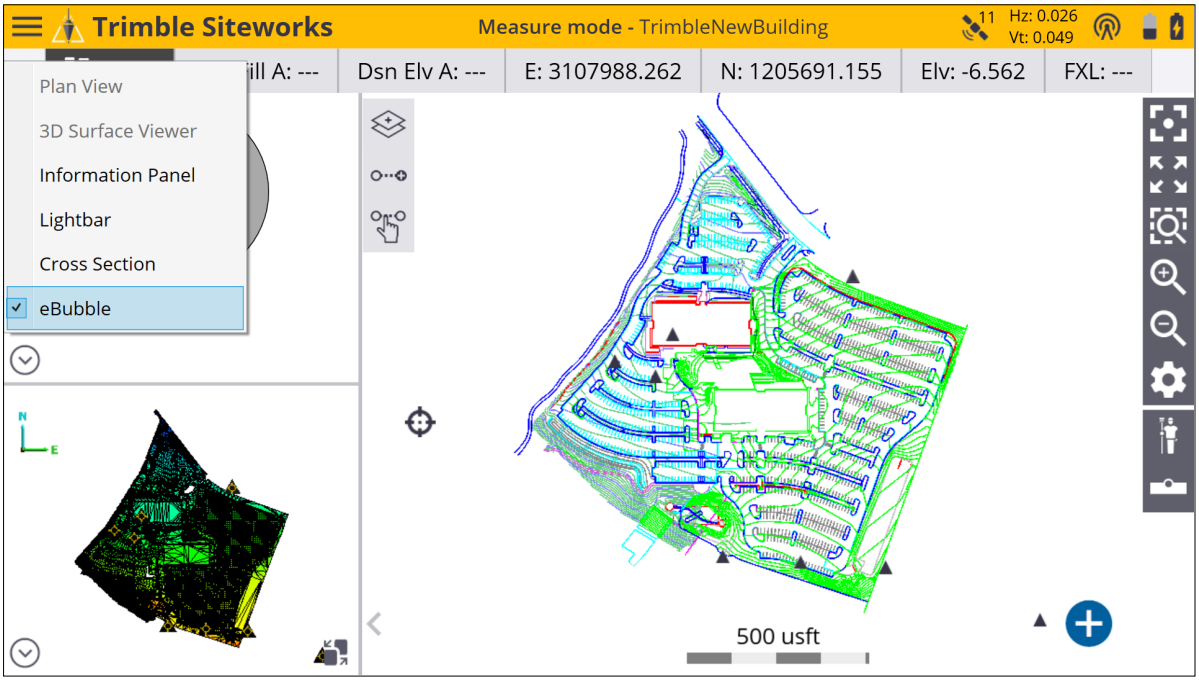
NOTE – A Cross Section is only displayed if road or alignment data is loaded in the current design. This feature is only enabled for users with the Roading module who are able to load alignment or road data.

Toggling between the panels

The main screen has three panels. The data in these panels can be set by selecting the main menu then **Settings**, **Map Display Settings** and selecting the **Panel Display** tab.

Alternatively, a quick way to select the data shown in the top and bottom panel is by clicking on the down arrow icon:

The required data can then be selected from the menu:

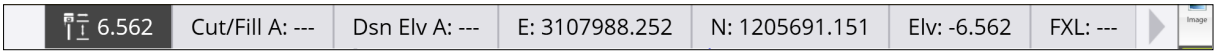


To display the data from the top and bottom side panels, select the **panel switch** icon in the bottom right of these small panels:

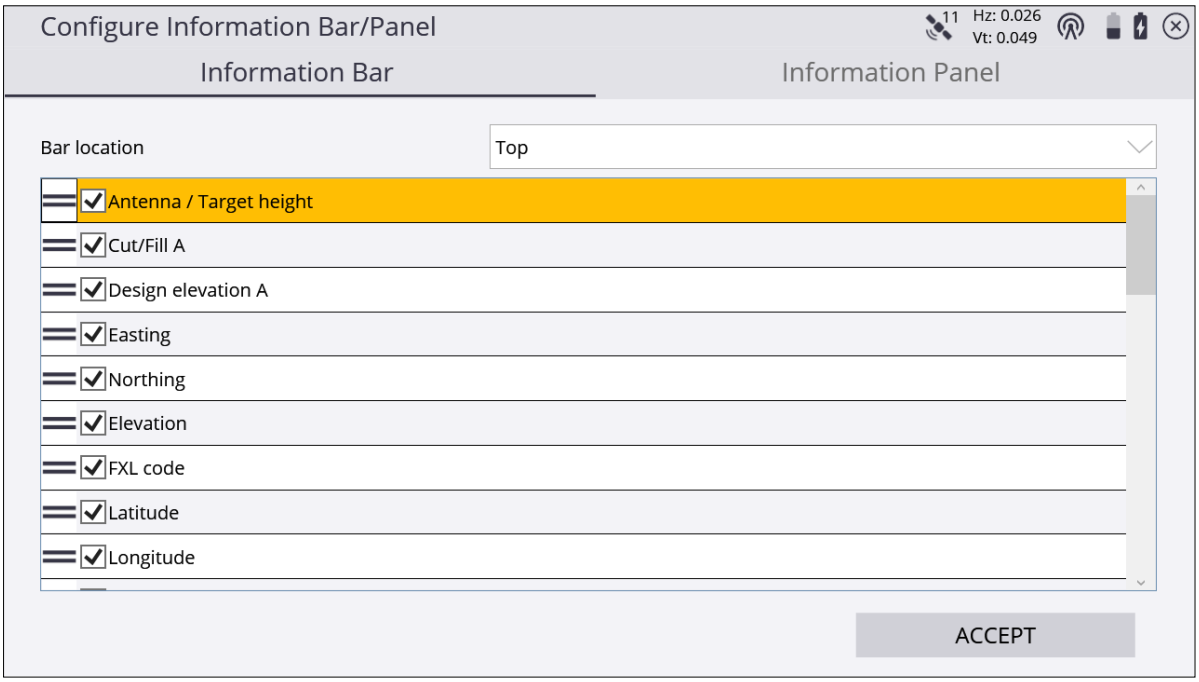
NOTE – The eBubble cannot be displayed in the main panel.

Info bar

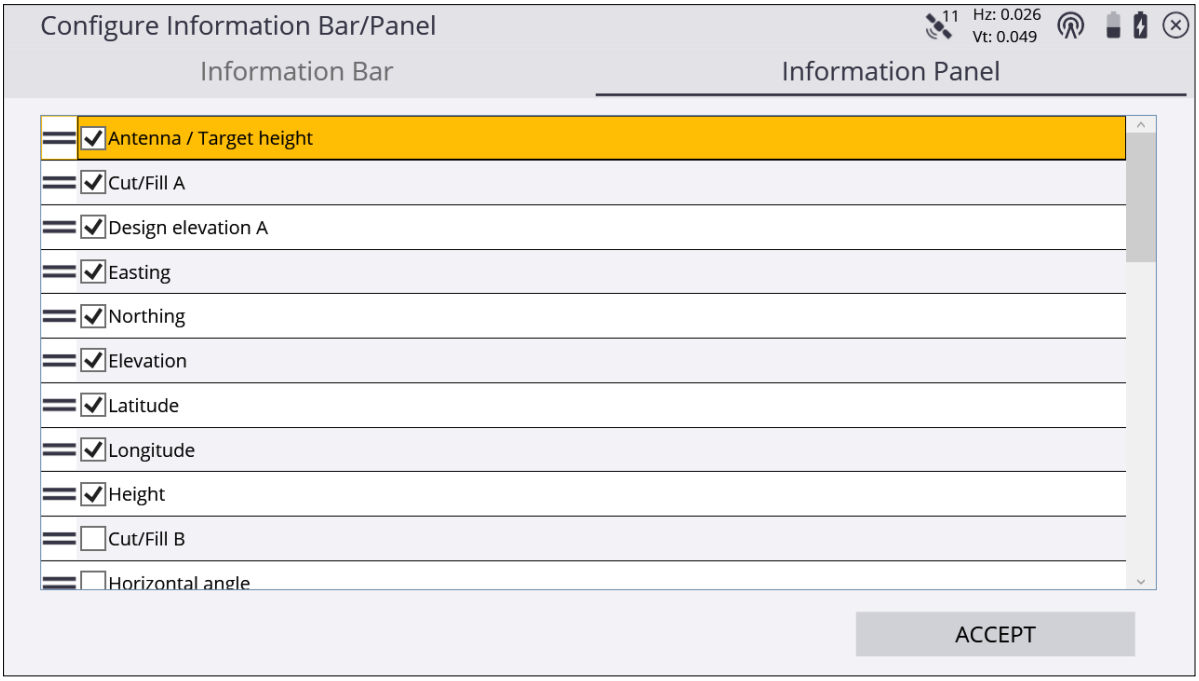
At the top of the screen is a bar that shows readings and values related to the current operation:



Use the arrows on the right and on the left to scroll through the different values which are currently enabled. You can also tap on the bar and "flick" through the different values. For each function, a predefined set of values is shown. You can modify the settings in the Home menu / Settings / Info Bar/Panel under the information bar tab



The Information Panel changes what is displayed when the information panel is selected to be displayed in one of the panels on the main screen.



Some fields in the information bar(for example, Antenna & Target Height, Stakeout line offset, and Surface offset) are "active" fields. By tapping on it, you can change the settings and shortcut in the **Settings** dialog of this value. The active fields are indicated by white text on dark gray background.

The following values are available; those indicated with an * are the “active” fields:

This value	Shows the...
Ant Ht *	currently applied antenna height for GNSS.
Tar. Ht*	currently applied target height for total station.
E	current Easting in the selected/applied coordinate system.
N	current Northing in the selected/applied coordinate system.
Elv	current Elevation in the selected/applied coordinate system.
Horizontal angle	current horizontal angle the instrument is shooting.
Vertical angle	current vertical angle the instrument is shooting.
Slope distance	current slope distance the instrument is shooting.
Lat	current latitude in WGS-84.
Long	current longitude in WGS-84.
Ht	current height in WGS-84.

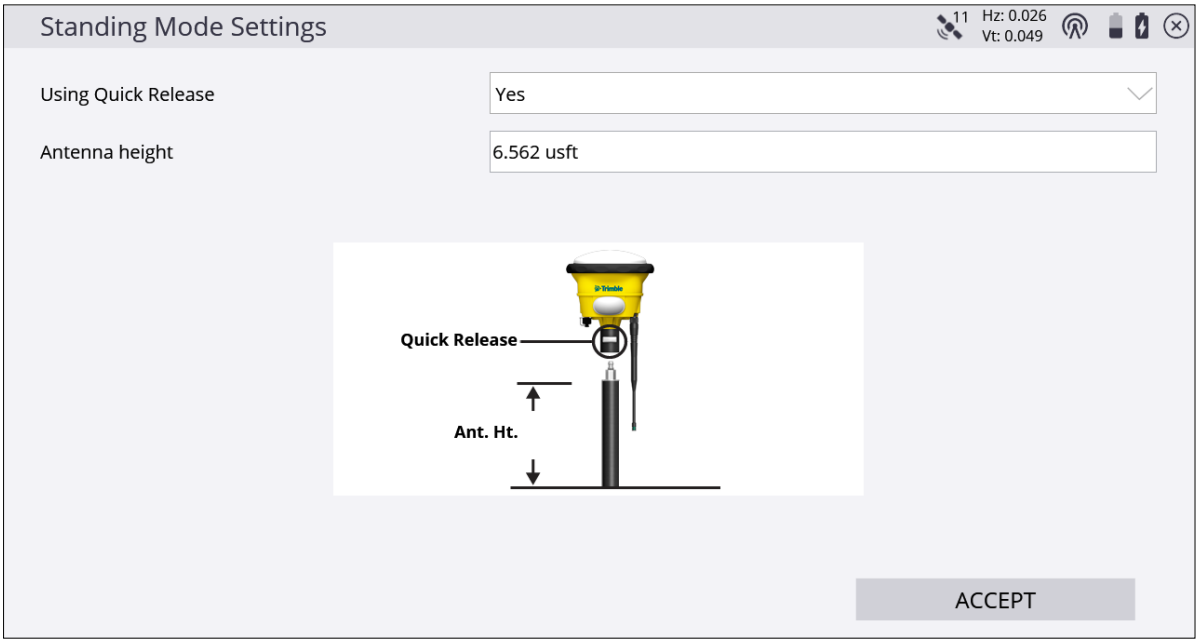
This value	Shows the...
Sta	current station to a selected road or alignment.
Off	current offset from a selected road or alignment.
Go	distance and direction guidance to a selected point or object.
Cut/Fill	cut/fill value to a selected design, road, or alignment.
Dsn Elv	elevation of the selected design, road, or alignment at the location of the positioning instrument.
Thickness	current vertical thickness of a layer from on the last layer.
R. Sta	current station to the selected reference alignment or line.
R. Off	current offset to the selected reference alignment or line.
dE	difference in East to a selected point or object.
dN	difference in North to a selected point or object.
dZ	difference in Elevation to a selected point or object.
Ahead/Back	difference in station to a selected point along the selected alignment.
Inward/Outward	difference in offset to a selected point relative to the selected alignment.
Feature 2 cut/fill	cut/fill value of the feature node that is created while staking a roadway feature using dual segments.
Feature 2 design elevation	design elevation of the second roadway feature created by the dual segments setting.
Dsn Sta	station to a selected point per design.
Stakeout line offset H*	currently-applied horizontal line offset.
Stakeout line offset V*	currently-applied vertical line offset.
Surface offset*	currently-applied surface offset.
Cut/Fill A	Cut/Fill values referenced to the primary surface.
Cut/Fill B	Cut/Fill values referenced to the secondary surface.
Design Elv A	design elevation referenced to the secondary surface.
Design Elv B	design elevation referenced to the secondary surface.

This value	Shows the...
Cut/Fill between surfaces	distance between the primary and secondary surface at this location.
Second surface settings*	surface selection and offset options for secondary surface.
FXL code	description for the selected feature code.

Antenna Height/Target Height

To change the antenna or target height, tap on the value in the info bar 

. This value is subtracted from each GPS elevation or elevation measured with a total station. If the antenna/target height is currently not displayed in the info bar, turn it on in the Home menu / Settings / Info Bar/Panel using the Information Bar tab.



Select whether a quick release is being used on compatible GPS receivers, but only enter the height to the bottom of the quick release as shown in the diagram. Siteworks automatically accounts for the height of the quick release and indicates its use on the map screen with an icon showing a quick release:

Each change of the antenna or target height and quick release use is stored in the log file, and can be viewed in the Work Order's task log, or by pressing the Home Menu/ Data Management / Log.

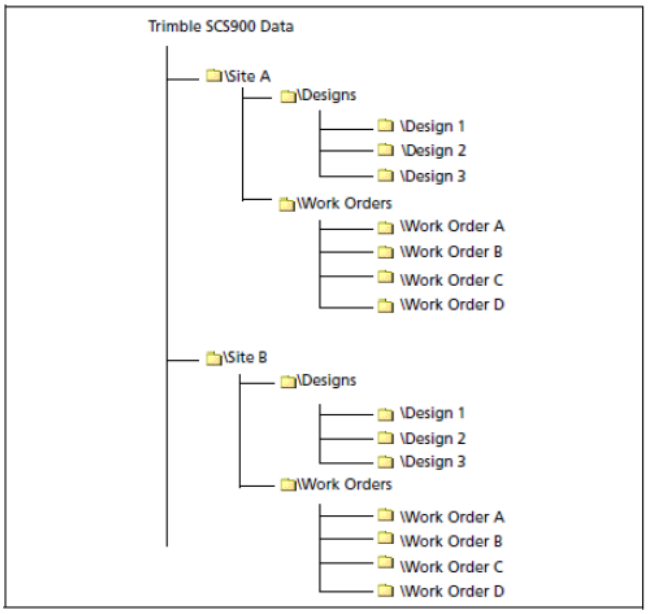
5

Data Management

Trimble recommends that you use the latest version of the Business Center – HCE software to prepare the data to be used with the Siteworks software. The Business Center – HCE software incorporates the capabilities of the SCS Data Manager and Report Utility for multiple jobsite/controller management and generating work order reports. The correct file folder structure is automatically created, and quality assurance tools are available to ensure that all controllers are using the most recent data.

Sites, designs, and work orders

All data for the software is stored in a top-level folder called Trimble SCS900 Data, and is organized in a precise structure. The file folder structure created on the office computer exactly mirrors the file folder structure on the controllers, which makes it easy to manage and archive data between the computer and the controllers. Data is organized by project. Within each project, data is divided into designs and work orders.



Work orders can be created directly in Siteworks or exported from the Business Center - HCE software. Creating a work order in Business Center - HCE allows for the assignment of the necessary design, surface offsets, measurement tolerances, instructions, coverage map grid size, and continuous measurement settings. Keeping data organized in work orders allows for easy import and data management within the Business Center - HCE software.

Information on a controller is arranged in the following levels:

Level	Description
Global	Global information is used at all sites. It includes lists of feature codes and Geoid files, and software information such as last connected site.
Project	Project information relates to all activities at the specified site. It includes control points, site calibration results, and background maps. Project information is always available.
Design	Inside each project, a main Designs folder holds individual design folders that contain design data pertaining to the project. Design data

Level	Description
	relates to a particular phase of construction. Data stored at this level includes foreground maps, stakeout data, and design surface models.
Work Order	Inside each project, a main Work Orders folder holds individual work order folders. This is where the measured data, and any exported data are stored.

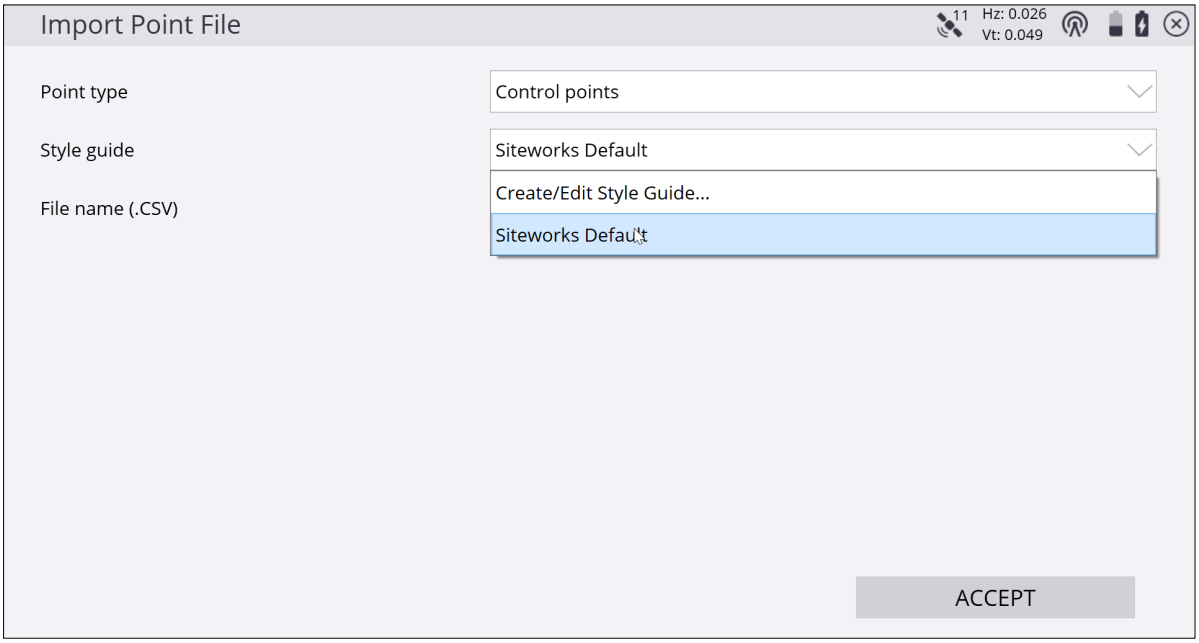
Importing and exporting point files with style guides

The Siteworks software enables you to import and export comma, tab, or semicolon delimited ASCII point files in the Siteworks default PNEZD, PENZD or in custom user-defined formats that are set by a style guide.

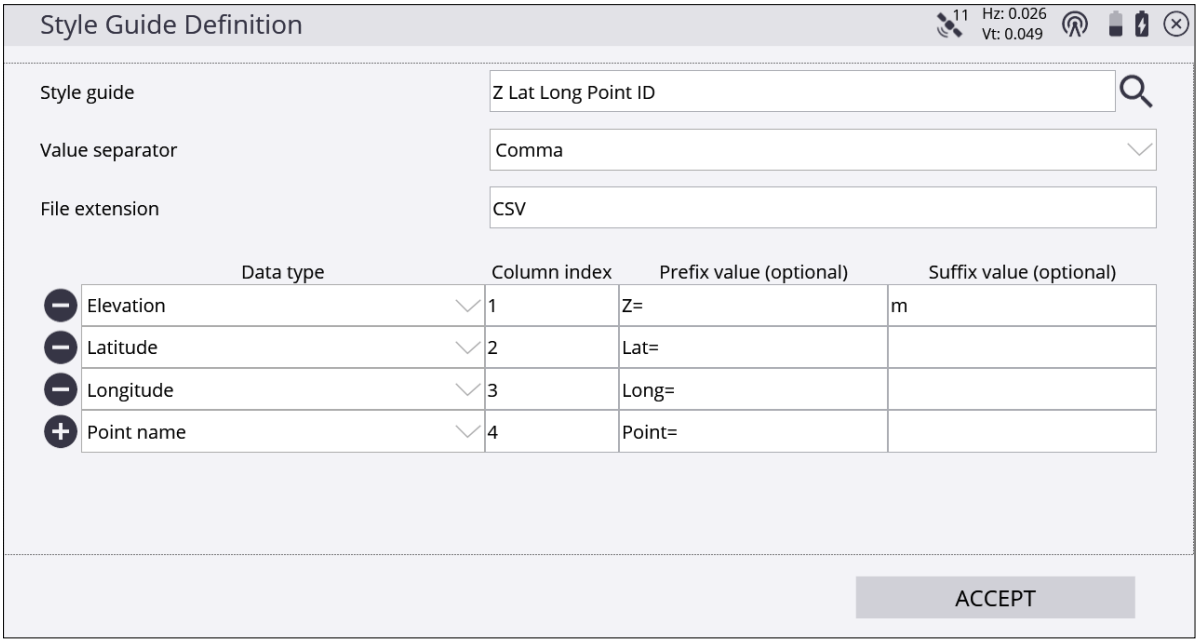
To import/export point files with formats other than the Siteworks default PNEZD or PENZD, you need to define a point file style guide.

Style guides are saved in a PointStyleGuides.jsn file in the Trimble SCS900 Data folder. This .jsn file is portable between field controllers and is synchronized during a Trimble Connected Community sync.

In each window where a point file can be selected there is an option to select the style guide or create/edit a new one. Current windows where a point file can be imported/exported include: Importing control points during the site creation process, Importing a point file from the **Data Management** menu, and Exporting Measured data via a Custom Point File. The Siteworks default style guide is the standard PNEZD or PENZD format. The coordinate order of the Easting and Northing during import/export using the standard Siteworks default format is set in the Site Creation process.



Select the Create/Edit Style Guide option. The **Style Guide Definition** screen appears:



Here you can configure various parameters of the file. The basics of style guides involve setting the style guide name, value separator (delimiter), file extension, if each column has a prefix and/or suffix, and then assigning a Siteworks data type to each column in the file.

Prefixes and suffixes can be defined by typing the optional value directly into the third and fourth column. During import, anything listed in the prefix or suffix box for that data type is ignored, and only the alphanumeric values shown before or after the prefix/suffix are imported. Exporting a point file using a prefix/suffix adds the respective prefix/suffix to the value being exported. The Siteworks software cannot automatically recognize what the data types are based on the prefix or suffix; that information must be set up in the style guide.

To configure a style guide:

1. Select the Create/Edit Style Guide option and enter a name for a new style guide. Alternatively, tap the magnifying glass to open an existing style guide to edit.
2. Select the value separator (comma, semicolon, or tab) that delimits the columns of data in the source file.
3. The file extension will be written to the exported file name during an export, and provides a means to filter by file type during file imports. Only those files with matching file extensions will be shown in the select file window during import.
4. To import files with different file extensions but otherwise with the same column format, select the All Files option in the **Type** screen.

5. To define the data type for each column select it from the drop-down list, and enter the column where it resides in the source file. If appropriate, enter the prefix and suffix values to be added.
6. To add an additional data type, tap **+**. To remove an existing data type, tap **-**.

NOTE – During a point import process, only the Point Name, Easting, Northing, Elevation, and Point Code are imported into the Siteworks software; all other fields in the file and associated style guide are ignored during import. An exported point file will contain all the attributes established and configured in the style guide. These attributes include: Antenna/Target Height, Date, Easting, Elevation, HA, Latitude, Longitude, Northing, PDOP, Point Code, Point Name, Slope Distance, Time, VA, and WGS Height. Also note that an exported file will contain column header names in the exported file.

Avoidance zone settings

You can use avoidance zones in the Siteworks software. To create the avoidance zone, use the Business Center - HCE software. In a CAD file, create closed polygons around the different zones and select them as avoidance zones in the site creation process. Business Center – HCE software will place the CAD file called *.avoid.dxf in the SCS900 Site folder.

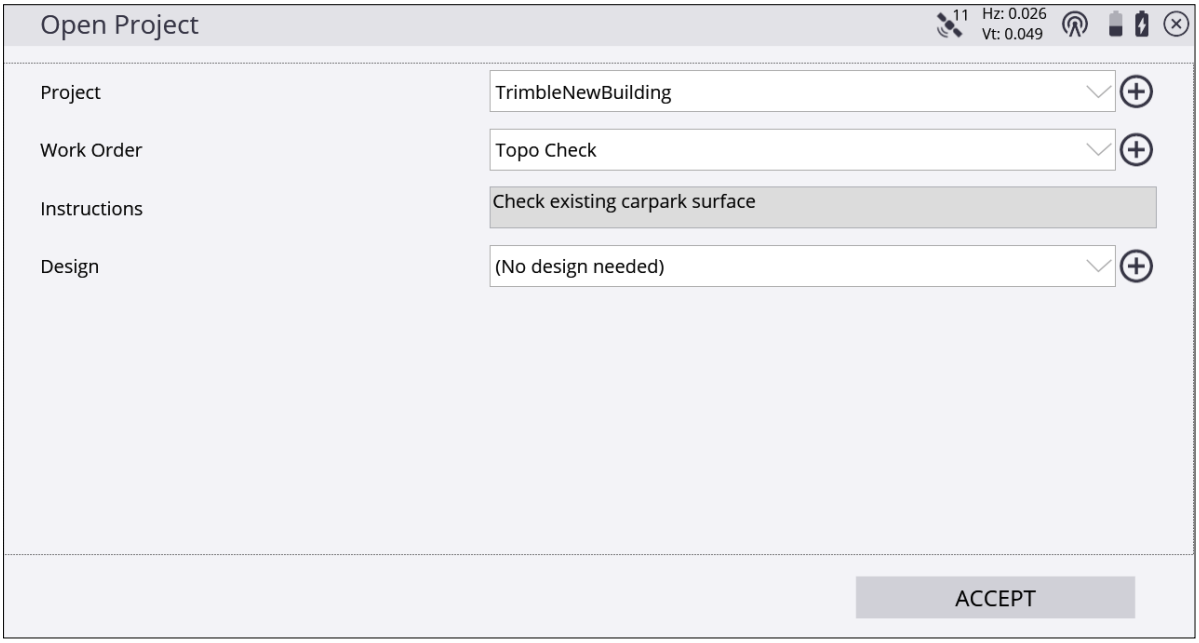
As you approach an avoidance zone, the software warns you audibly and visually:

- Yellow = Inside the avoidance zone tolerance area
- Red = Inside the avoidance zone

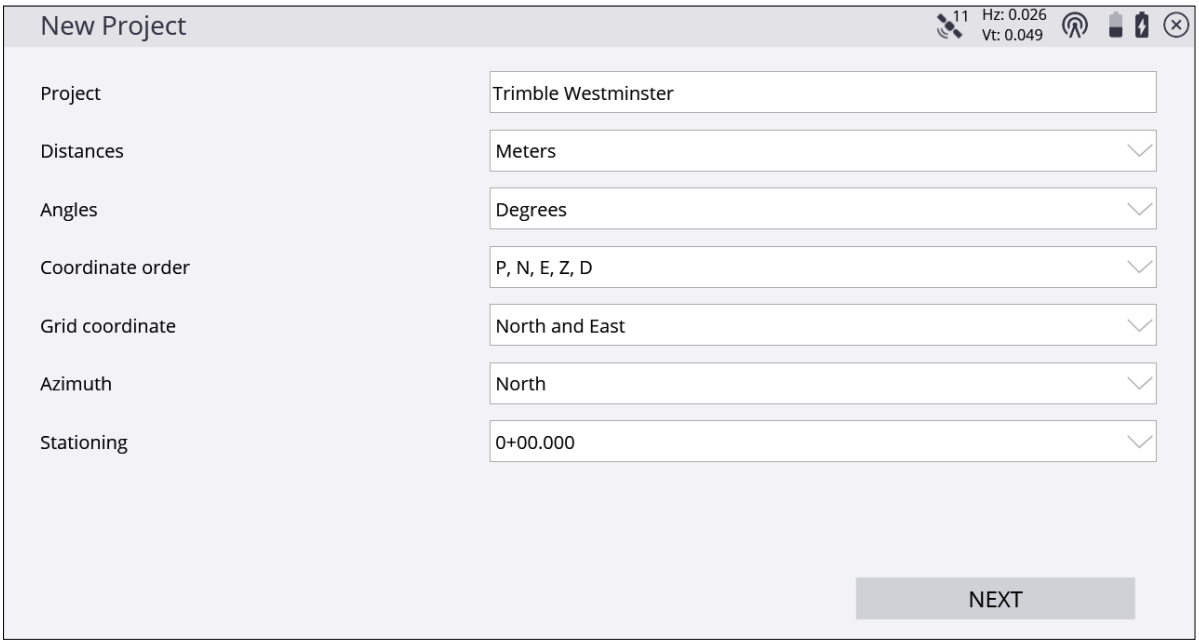
To adjust the tolerance for the avoidance zone, select the / Settings / Avoidance Zone.

Creating and opening a work order in the field

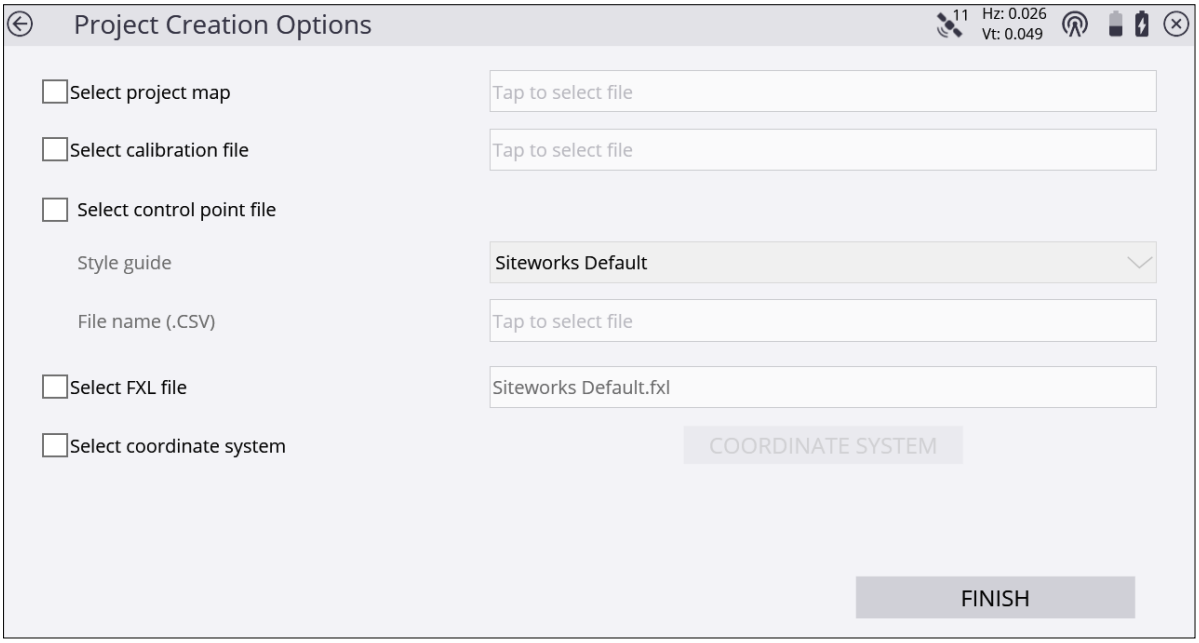
From the **Home** menu, tap **Project Setup** and then **Change Project**. After starting the software, you can select an existing project and work order to start with:



For each of these options, you can either decide to open an existing project, design, and work order, or create a new one. Tapping the plus symbol ⊕ on the right side opens a screen with a number of fields where you can define the settings and add data files from a USB drive or from data already stored on the controller.

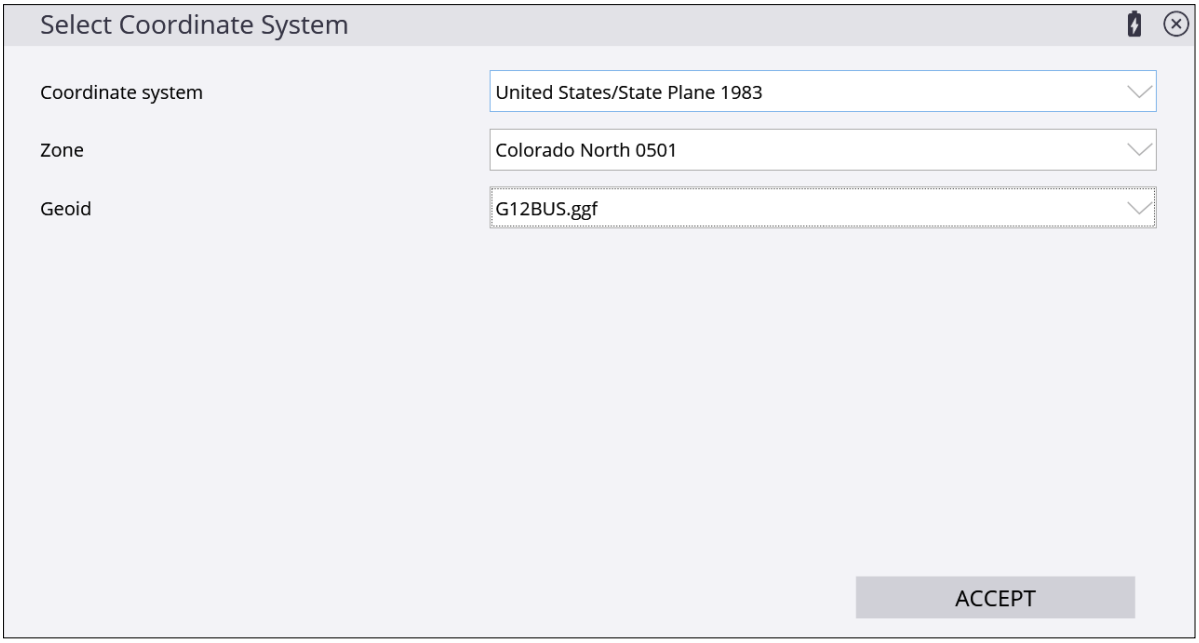


⚠ CAUTION – Before taking a measurement or associating a design with the site, ensure that the distance units are correctly set. All files that relate to a single site must be stored and operated with the same units. Once a measurement is taken, or a design is selected, you cannot change the units.

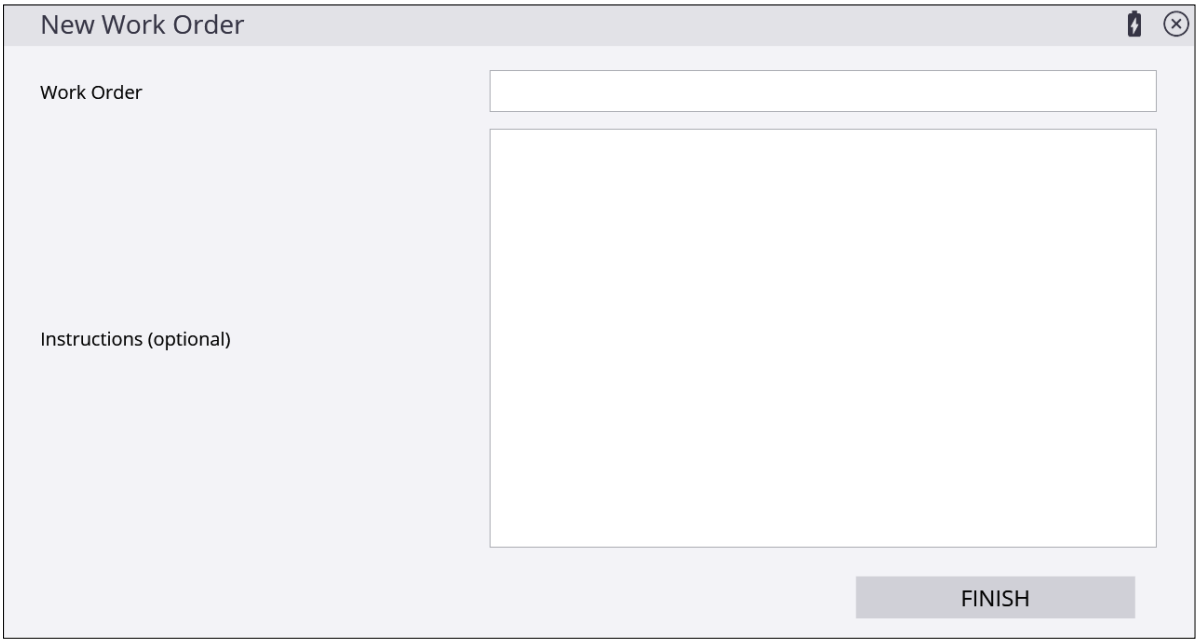


When creating a site on the controller, you can import or measure a site calibration or use a published coordinate system from the coordinate system manager. After you select the **Select coordinate system** check box and tap **Coordinate System**, the software will list all supported coordinate systems.

To use a geoid for the first time, it must be exported from the Business Center - HCE software, and stored in the Trimble GeoData folder on the controller.



A work order can include instructions for the person in the field, explaining what tasks to carry out. Work order instructions entered in the Business Center – HCE system will appear in this screen when selecting the work order. When creating a work order in the field on the controller, instructions can be entered in the instruction box:



Work orders should have a meaningful name to make it easy to identify them when multiple work orders are created for a particular project.

Measured points are saved within individual work orders. For example, if you are staking a building pad, the saved stakeout values are saved into the work order folder. Then when

measuring topo on the same site but in a different work order, the measured topo points will be in that different work order. Also, any data exported via the **Home/ Data Management** menu is placed in an Output folder in the associated work order folder to make it easier to separate work tasks and keep the associated data compartmentalized.

Connected Community service

The Connected Community service includes the following services:

- Wireless Data Synchronization to synchronize Siteworks data with the data stored on the Connected Community website.
- IBSS (Internet Base Station Service) web service to receive base station corrections from the local base station through the Internet.

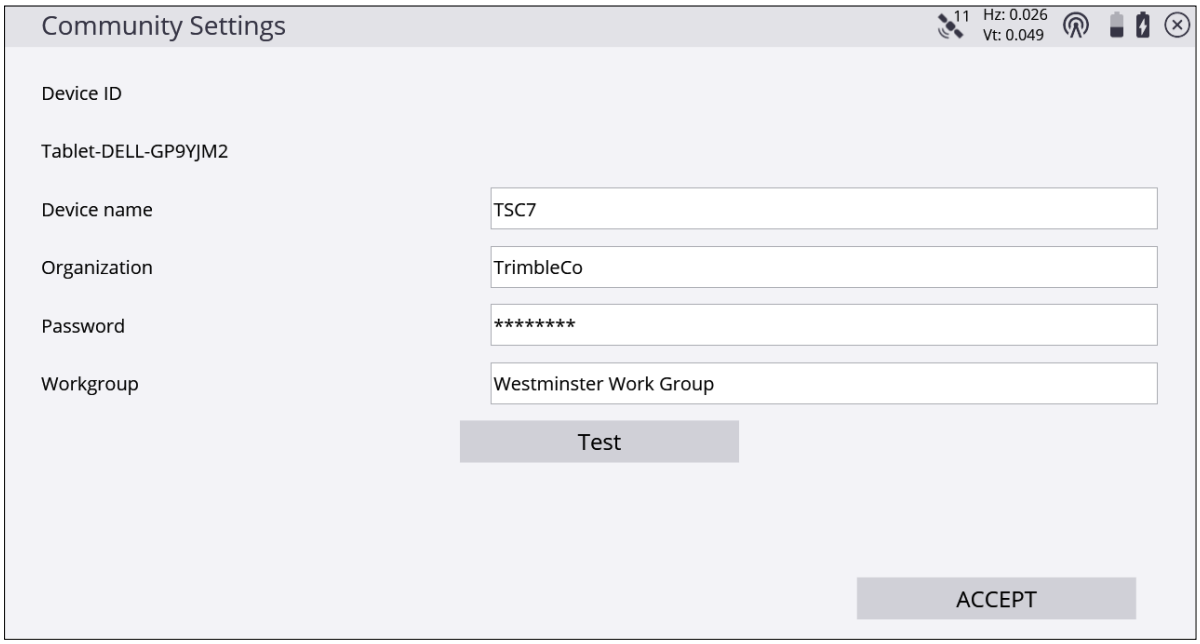
All these services are tied to the device ID that you need to purchase for the controller with a monthly TCC subscription.

Registering the controller

To use the controller with Trimble Connected Community (TCC) services, you must register the controller online in the Trimble Connected Community Device Manager of the user organization.

1. In the Siteworks software **Home menu**, select **Data Management**.
2. Tap **Community Settings** 

.
3. Enter the credentials and then tap **ACCEPT**. The Device ID is hardcoded and set to the serial number of the device. The Device Name is set up the first time the credentials are entered and cannot be changed once set; that will be the name of the folder for the device's data that appears on the TCC.



You only need to do this once. You can also enter a work group for Wireless Data Sync to group multiple controllers of a company in a certain structure.

Wireless Data Sync

The Wireless Data Sync option enables Siteworks data to be synchronized with data stored on the Trimble Connected Community website, which eliminates the requirement to physically move data to and from the field by plugging the controller into a computer or using USB sticks. Data can be synced right from the field as long as an Internet connection is available to the controller via Wi-Fi or a cellular modem.

To manage the synchronization process, the following set of rules controls the dataflow to and from the Trimble Connected Community website.

File Type	Currently on the Connected Community	Currently on the controller	Action
Work Order	✓	✗	Download to the controller
	✗	✓	Upload to the Trimble Connected Community service
	✓	✓	Upload to the Trimble Connected Community service if the file size is different
Design Data	✓	✗	Download to the controller
	✗	✓	Upload to the Trimble Connected Community service
	✓	✓	Download if the file size is different
Site	✓	✗	Download to the controller
	✗	✓	Upload to the Trimble Connected Community service
	✓	✓	Download if file size is different

File Type	Currently on the Connected Community	Currently on the controller	Action
Calibration File (*.DC & *.CAL)	✓	✗	Download to the controller
	✗	✓	Upload to the Trimble Connected Community service *NOTE – If data that has been previously synced is deleted from TCC but remains on the controller, you will be prompted to delete it from the controller.*
	✓	✓	Ask when the file size is different
Field Control Point File (*.field.csv)	✓	✗	Download to the controller
	✗	✓	Upload to the Trimble Connected Service *NOTE – If data that has been previously synced is deleted from TCC but remains on the controller, you will be prompted to delete it from the controller.*
	✓	✓	Ask when the file size is different
Office Control Point File (*.office.csv)	✓	✗	Download to the controller
	✗	✓	Upload to the Trimble Connected Community service *NOTE – If data that has been previously synced is deleted from TCC but remains on the controller, then you will be prompted to delete it from the controller.*
	✓	✓	Download if the file size is different
FXL File	✓	✗	Download to the controller
	✗	✓	Upload to the Trimble Connected Community service
	✓	✓	Ask when the file size is different

File Type	Currently on the Connected Community	Currently on the controller	Action
Report.txt, Tasklog.txt	✓	✗	Download to the controller
	✗	✓	Upload to the Trimble Connected Community service
	✓	✓	Upload if the file size is different
Site.ini, Site.xml, Site.xml.schema	✓	✗	Download to the controller
	✗	✓	Upload to the Trimble Connected Community service
	✓	✓	Do nothing
Trimble GeoData	✓	✗	Download to the controller
	✗	✓	Upload to the Trimble Connected Community service
	✓	✓	Download if the file size is different

The software stores the designs and work order names that have been synchronized in a history file with every synchronization. If designs or work orders are deleted from the Trimble Connected Community service since the last synchronization, the Siteworks software will also delete the files from the controller and then proceed with normal synchronization rules from above thereafter. Deleted work order data from the controller is stored in an Archive folder on the TCC servers. If control point files and DC files have been deleted from TCC since the last synchronization, then the Siteworks software prompts you to delete the files from the device. Files selected for deletion will also be placed in an Archive folder on TCC to preserve a record of them.

The software facilitates two Control Point files in the site folder: a Field Control Point file, where all control points measured or typed in the field are stored; and one or more Office Control Point files. The following file naming conventions must be used:

- Office Control Point File: [filename].office.csv
- Field Control Point File: [filename].field.csv

The Office Control Point file cannot be edited in the field, however the Field Control Point file can. Using the Business Center – HCE software, the data manager can take the control

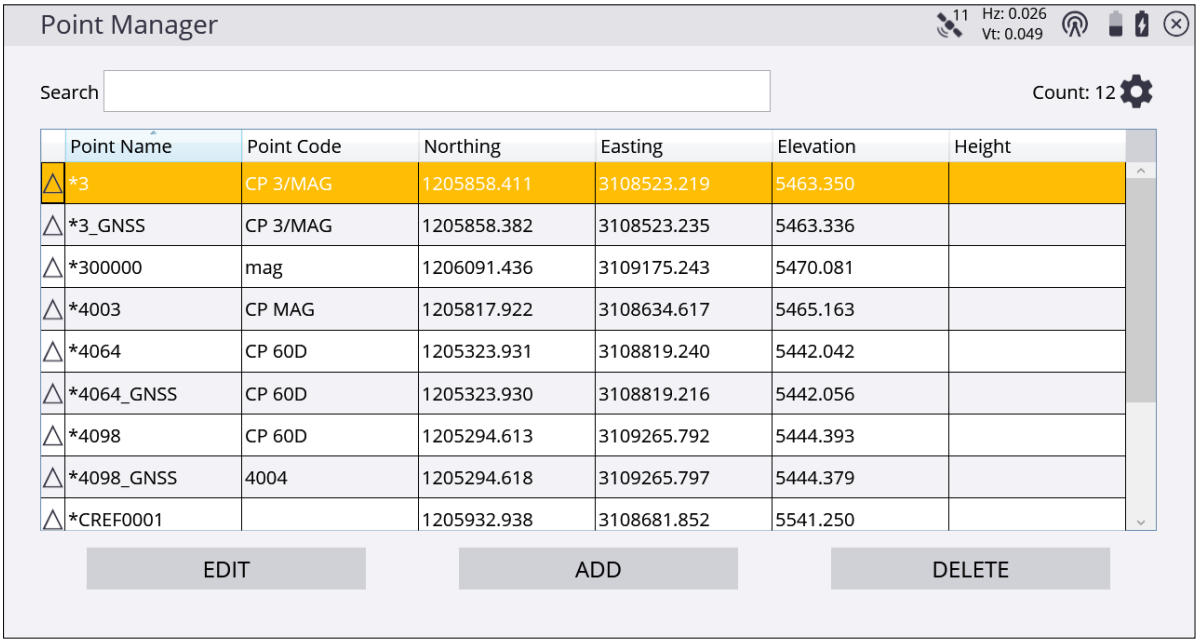
points from the Field Control Point file of one controller, move it to the Office Control Point file and then push the new Office Control Point file out to other controllers.

When a site is opened for the first time and the software cannot find a Field Control Point file, the software converts the name of the existing Control Point file selected during the site creation process, and applies the naming convention for the Field Control Point file. From that point forward, the software will ignore all other CSV files in the Site folder that are not tagged office or field. The synchronization with TCC will only include Control Point files that are marked as Office or Field Control Point files. The synchronization rules for the Office and Field Control Point file from above apply.

Point Manager

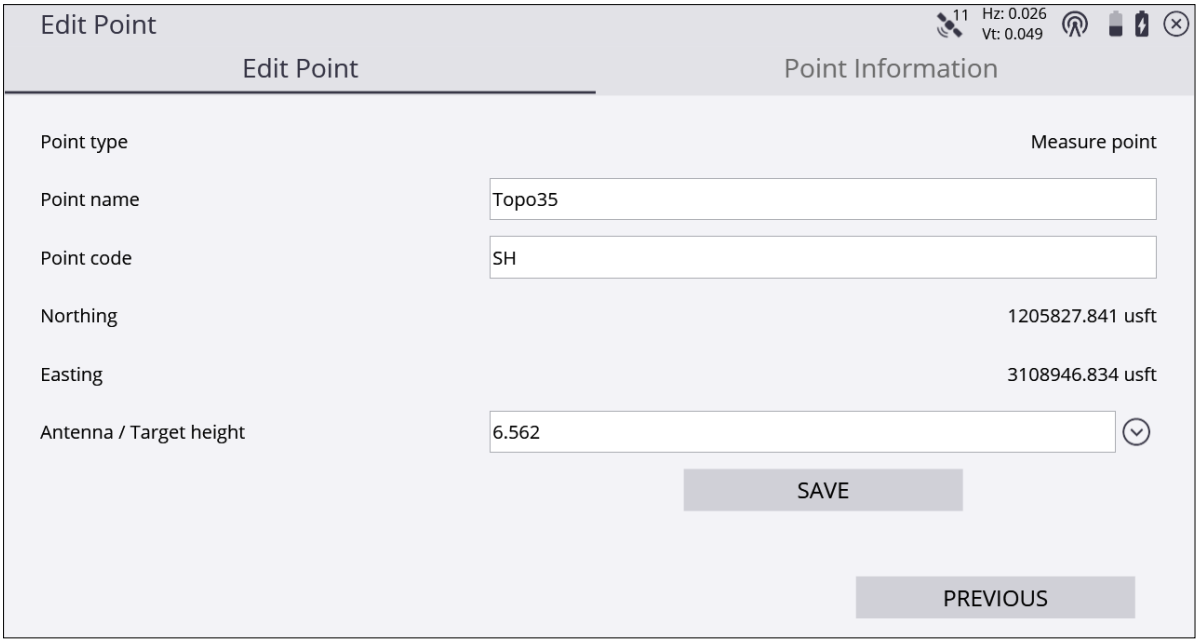
To access the Point Manager, select **Home** / **Data Management** / **Point Manager**. Options will appear to select either the point manager, view/ edit control points, or to import control file. Select Point Manager. Use this feature to retrieve a list of all points in the currently loaded work order. Tap **Edit**, **Add**, or **Delete** to make changes to an existing point, add a new point, or to completely delete a point.

Columns can be sorted by clicking on the column name. Use the search box at the top of the window to search by **Point Name** and **Point Code** fields. Point types can be filtered to display only certain point types by selecting  . Office control points, denoted by the * cannot be changed.



Tap **Add** to enter 1D, 2D, and 3D control points, and 2D and 3D Stakeout points.

To edit points, select a point and then tap **Edit**. You can change the point name, point code, elevation, and antenna/target height for measured points. Click on the down arrow to the right of the data field to switch between elevation and antenna/target height. Northing and Easting values of measured points cannot be changed, but may be changed for non-measured points that were added by key-in. If an FXL file and the Advanced Measurement Module are active, you can assign an FXL code to the point. FXL attributes cannot currently be edited for existing points; but they can be replaced with new information by tapping **Select FXL Code**.



Clicking **Previous** or **Next** takes you to the previous or next point in the list according to the sorting set in the list view. Unsaved changes will be prompted for saving before switching points.

Information about the point can be viewed by selecting the **Point Information** tab.

6

Measurement Workflows

The Siteworks software is a site measurement tool that enables you to monitor earthworks and surface finishing operations. It enables construction contractors to measure material volumes, monitor and measure grade elevations relative to the site design, determine laid material thicknesses, and to perform site measurement tasks such as measuring points, lines, and surfaces.

Displaying cut/fill

A valid design containing a surface model must be loaded via the **Project Setup** menu for cut and fill values to be displayed. If the software is not in Measure mode, tap the **Home** button and then tap **Measure**. Walk anywhere on the design surface and view the current cut/fill to the design elevation on the Info bar, page 46. The lightbar can be displayed in any of the panels by clicking the down arrow in the bottom or top side panel.

This lightbar indicates whether the surface is in cut, fill, or on grade:

- Blue = fill (below design elevation)
- Red = cut (above design elevation)
- Green = on grade

The lightbar can also be turned off/on in the **Panel Display** tab of the Map Options by selecting/deselecting the **Lightbar** check box. The lightbar will blink when approaching within approximately 0.33 m or 1 ft of grade elevation.

Audio indicators also will also sound when within 0.33 m or 1 ft of grade:

- A steady tone = on grade
- A slow beeping = below grade
- A fast beeping = above grade

The lightbar sound can be turned off/on in the **Design** tab of the Map Options by selecting/deselecting the **Lightbar sound** check box.

Checking a grade/elevation

Measure a surface point at a location where you want to view and record the difference in elevation between the design surface and the actual ground. As you move around, the values in the info bar at the top of the screen update with current values.

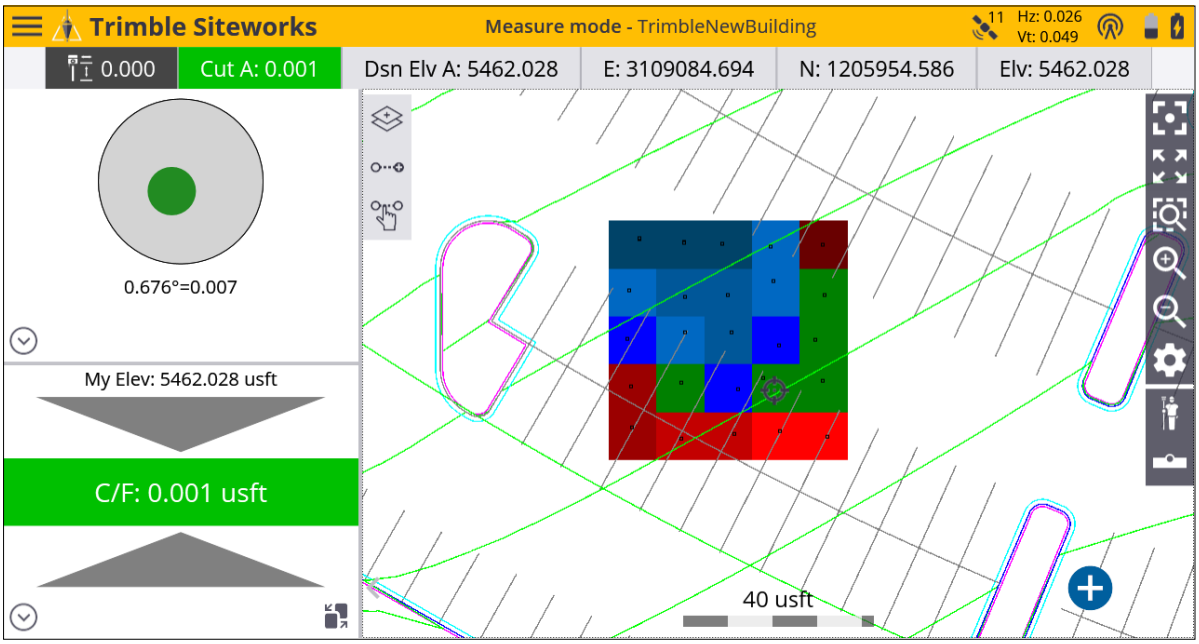
1. If not in Measure mode, tap the **Home** button and then tap **Measure**.
2. Tap the Measure button to record a surface point and the cut/fill value at that location.

The software draws a colored grid box of the size that you specify in the **Settings** / **Map Display Settings** / **Measure** tab around every recorded point so you can easily view where data is missing, and determine areas of cut or fill.

Once a point is recorded, a colored box appears around it, showing it as in tolerance (green), cut required (red), or fill required (blue). The colors are shaded depending on how far away from grade they are.

To change the cut/fill tolerances:

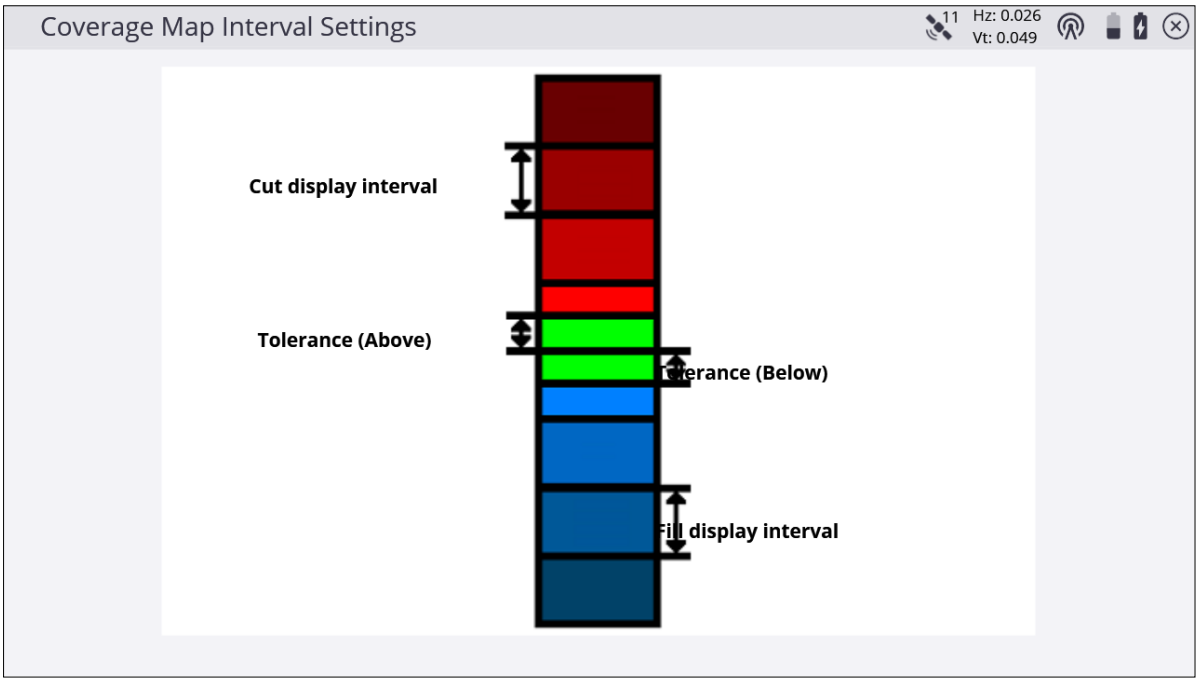
1. Tap the and then tap **Settings** / **Measure**.
2. Enter the required tolerances and then tap **ACCEPT**.



TIP – If a gray box appears, tap the zoom window icon and draw a box around the area of the gray box. Gray boxes appear when the map is zoomed out too far to see the colored boxes at the specified resolution.

If no boxes appear, tap  from the toolbar on the right. Ensure that the Coverage Grid check box and the Cut/Fill: Measured option are selected. You can also change the grid size. Note that if a too small grid size is entered, that will result in more than 600,000 grid cells being created over the site's extents, then no grid cells will appear. In this case, increase the grid size.

Grid cut/fill color display values are graduated into four shades each of blue and red. These cut/fill display color intervals are set through the Home menu / Settings / Measure settings as shown below.



The Tolerance (Above/Below) value is how far above or below the design surface elevation the measured point can be to plot in a green, within tolerance color. The Cut/Fill display interval is the distance over which one color shade will be plotted. Different values can be set individually for the Cut and Fill display intervals, along with different individual Tolerance above and below values.

Checking material thickness

The typical procedure for checking a material thickness is:

1. Measure the existing surface before laying the material.
2. Save the measured surface as a design.
3. Create a new work order and then load the saved design as the active design.
4. Lay the new material.
5. Check the material thickness by measuring points on the laid material, after entering a surface offset value equal to the required thickness.

If the current material thickness is too thin, a blue square appears to show that more "fill" material is required. If the current material thickness is too thick, a red square appears to show that material is required to be "cut" away. If the current material thickness is within a specified tolerance, a green square appears to show that no action is required.

1. If not in Measure mode, tap the **Home** button and then tap **Measure**.
2. Tap the **Home** button and then select **Settings** / **Measure**.
3. Ensure that the measurement offset is enabled in this menu (you can change tolerances here too).
4. Select the **measure type** icon in the main panel  and enter the required thickness as a vertical offset. Tap **ACCEPT**.
5. Tap **Measure**  to record a point and the cut/fill value at that location.

As you move around, the values in the info bar at the top of the screen update; the thickness of the material is shown in the Thickness box. The Thickness in the info bar can be activated, if it is not displayed, by pressing the **Home Menu** / **Settings** / **Info Bar/Panel**.

Once a point is recorded, a colored box appears around it showing whether it is within the tolerance range or whether more or less material is required.

TIP – If a gray box appears, tap the zoom window icon and draw a box around the area of the gray box. Gray boxes appear when the map is zoomed out too far to see the colored boxes at the specified resolution.

If no boxes appear, tap  from the toolbar on the right. Ensure that the **Coverage Grid** check box and the **Cut/Fill: Measured** option are selected. You can also change the grid size.

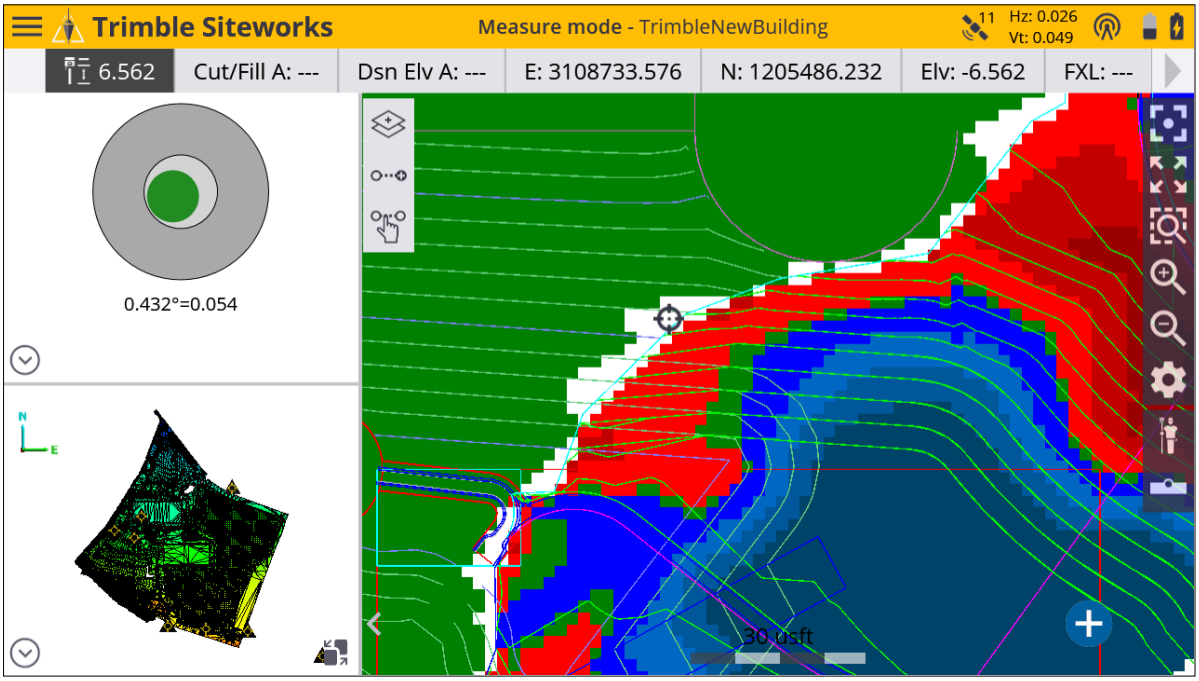
Note that if a small grid size is entered that will result in more than 600,000 grid cells being created over the site's extents, then no grid cells will appear. In this case, increase the grid size.

Checking cut/fill between two saved surfaces

It is possible to produce a grid display of the cut/fill between two saved design surfaces.

Tap  to open the **Map Options** screen and select the **Coverage Grid** option of Cut/Fill: Surface A-B. Surface A is the primary design surface loaded in the **Open Project** menu, and Surface B is selected via the / **Settings** / **Second Surface**. This displays colored cut/fill grid cells between the two surfaces using the color settings set via the / **Settings** / **Measure**, **Tolerance Above/Below** and **Cut/Fill Display Interval**.

If too small a grid size is entered, you are notified of the minimum grid size to produce a display grid.

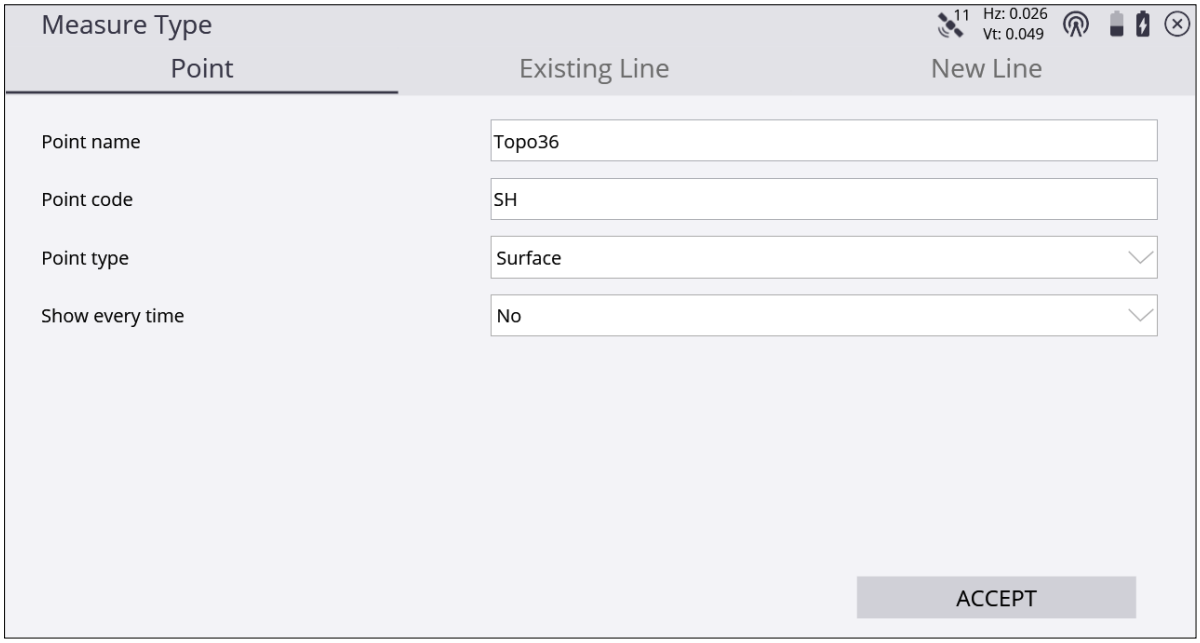


TIP – If a gray box appears, tap the zoom window icon and draw a box around the area of the gray box. Gray boxes appear when the map is zoomed out too far to see the colored boxes at the specified resolution.

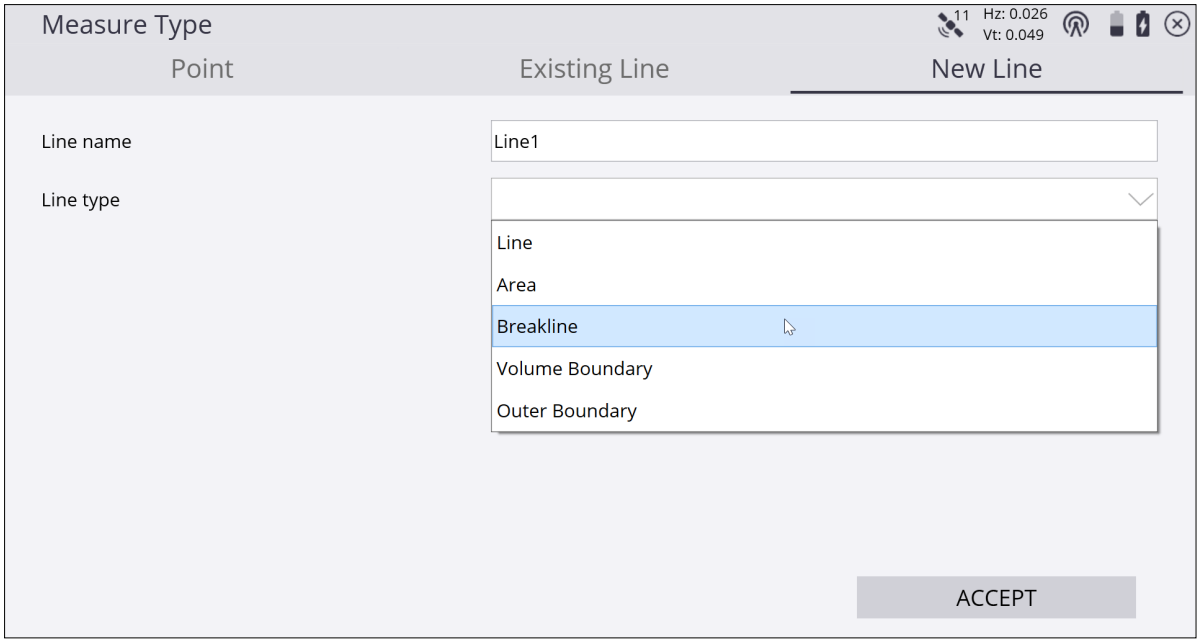
Measuring a surface or a feature

1. If the software is not in Measure mode, tap **Home** and then tap **Measure**.

2. To open the **Measure Type** settings screen, tap the icon highlighted above on the right hand side of the main panel to choose between point, line, surface, and non-surface feature to be measured:



Lines are measured by selecting one of the Line tabs for New Lines or Existing Lines. Selecting **Existing Line** enables the selection of an existing line from a table and the continuation of measuring that line from the last measured end point along it. Selecting **New Line** requires entering a Line Name, and choosing what kind of line type it is. Line types that are included in the DTM surface measurement are Breaklines, Volume Boundaries, and Outer Boundaries. Line and Area are not included in the DTM surface measurement.



3. You can also enter a point name (will be automatically incremented) and optional point code. The status bar icon changes depending on what kind of point or line you choose to measure:

Icon		Definition
	Surface Point	Elevation is used to create a terrain model.
	Feature Point	Elevation is not used to create a terrain model.
	Feature Line or Area	Elevation is not used to create terrain model.
	Breakline, Volume Boundary, or Outer Boundary	Elevation is used to create a terrain model.

To create an outer boundary, volume boundary, or surface points to add to an existing line, select the correct line type. Once a surface is measured, you can save the surface as a design and then perform a material thickness check. See Checking material thickness, page 71.

To save the measured surface as a design:

1. From the **Home** menu, tap **Data Management**.
2. Tap **Surface as Design** 

and enter a design name.
3. Choose whether to include measured, design, or no linework.
4. Select if you wish to merge the measured surface with the currently selected design surface.
5. Tap **ACCEPT** to export the measured surface as a design.

EZ Level command

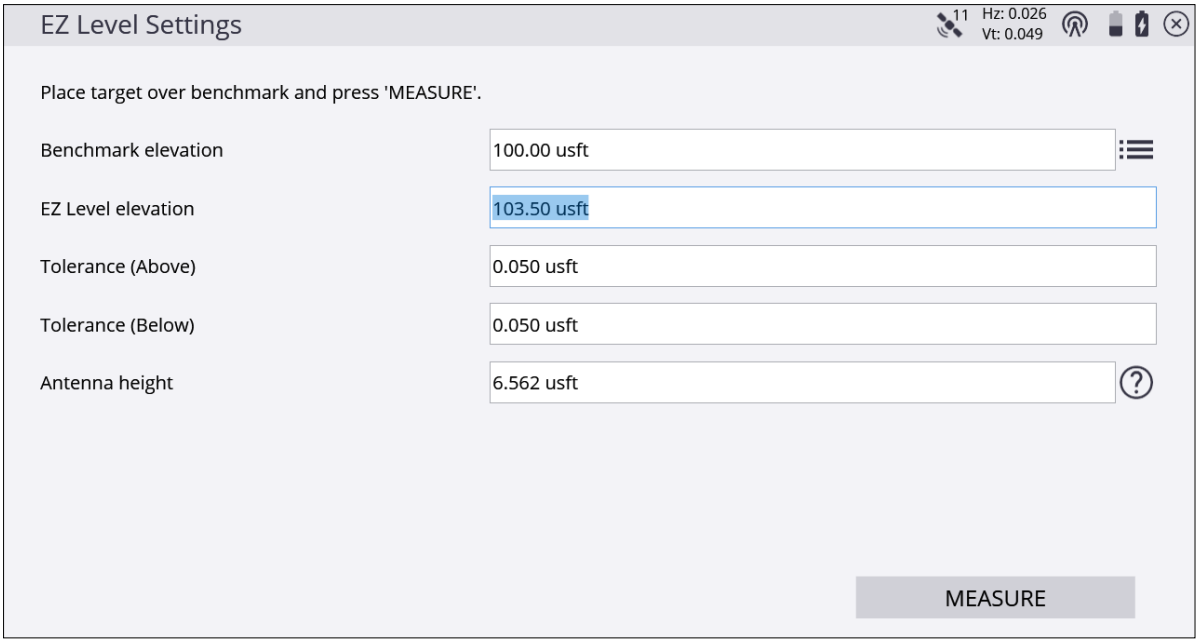
This feature provides a simple and easy way to graphically display elevations on a site relative to an arbitrary benchmark that is measured as part of the EZ Level workflow. This is a display and indicate-only feature and does not store any points or measurements. It is available for both GNSS and total stations. It does not require a site to be calibrated for GNSS use, or a total station to be set up on a known point. The intent of this feature is to provide a laser level-like workflow to quickly and accurately determine cut/fill values from an arbitrary EZ Level elevation relative to a measured benchmark elevation.

To open the EZ Level feature, tap the **Measure Mode** icon on the right side of the main panel or tap the **Measure** and select **EZ Level**:

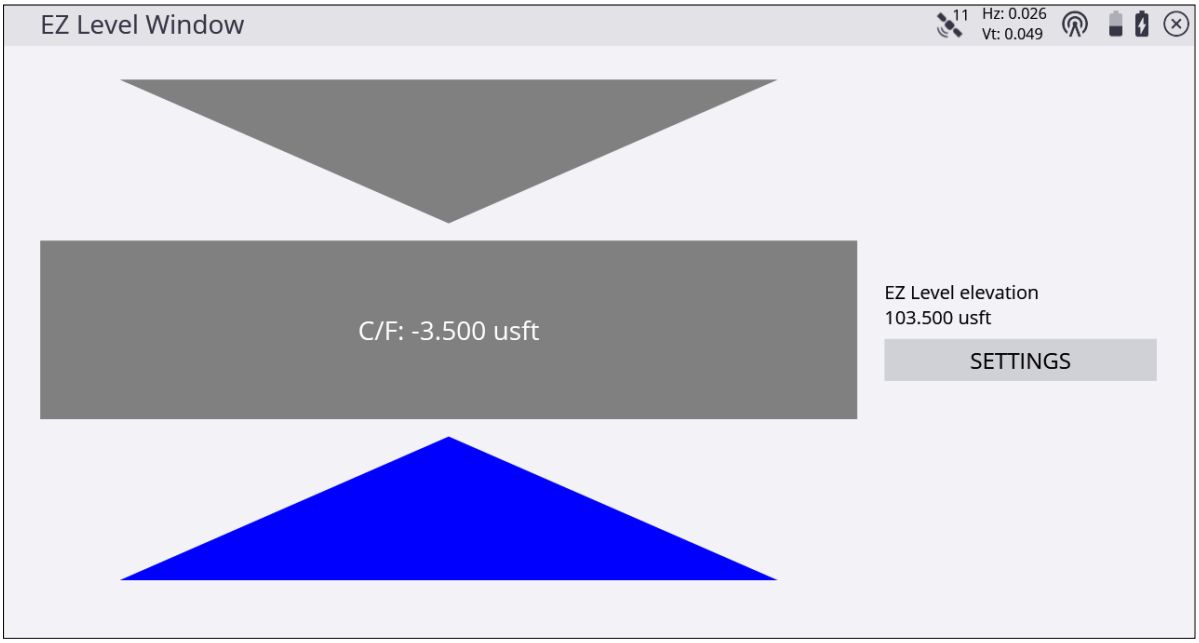
And then select **EZ Level**:

Alternatively, tap the **Home** menu / **Measure** and then select **EZ Level**.

Once in the **EZ Level Settings** screen, enter a benchmark elevation which is the reference elevation that you wish to measure. Enter an arbitrary value for the benchmark elevation or select the elevation of a control point from a list of control points by tapping the list icon:



The EZ Level elevation is the elevation value that is used to display the cut/fill values to on the **EZ Level** screen, and is relative to the arbitrary benchmark elevation. For example, if there is a benchmark at elevation 100, and you want to determine cut/fill values for a finished floor elevation of 103.5, you would place the rod on the benchmark at 100, enter **103.5** for the EZ Level Elevation, and then measure the benchmark. On-screen values then indicate the cut/fill relative to the 103.5 EZ Level elevation. The current elevation is displayed in the center of the screen, between the two arrows, and the cut/fill values to get to the EZ Level elevation are displayed on the upper or lower arrows respectively.



To re-enter the settings window to change the EZ Level elevation and/or measure a new benchmark elevation, tap SETTINGS.

Measuring with feature codes

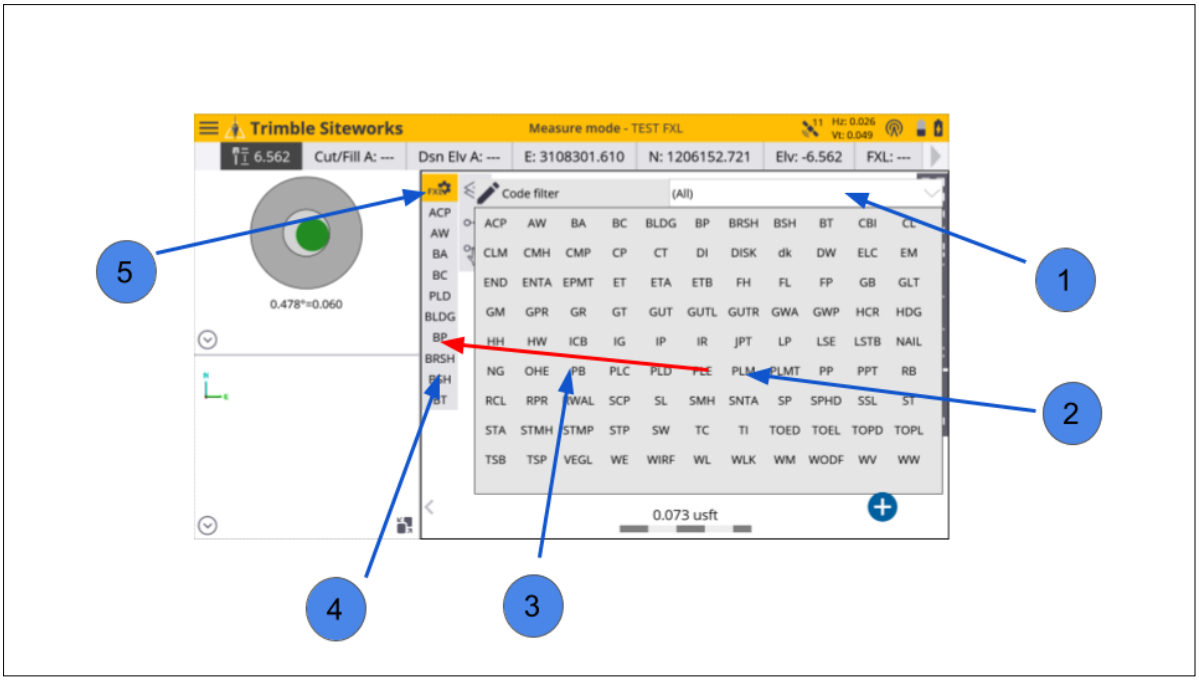
NOTE – The Advanced Measurement module is required.

The software can use feature codes to record data on site. Create and customize the feature code library using the Feature Definition Manager of the Trimble Business Center - HCE software.

The feature code defines if a point, line, or breakline is measured. The following classes of feature codes are available:

Feature code class	Definition
	Point feature, but no surface feature.
	Line feature, but no surface feature.
	Point feature and surface point.
	Line feature and breakline.
	Feature contains optional or compulsory attributes.

The currently selected feature code and its class can be identified in the status bar. To select and manage feature codes in the field, tap  .



Select this option...	to...
①	filter by group or category.
②	select a feature code.
③	drag code to the Quick Select list.
④	select from the Quick Select list.
⑤	toggle to the Grid view.

To select a feature code, either tap on one of the buttons in the Quick Select list or select a feature code from the Grid view. The Grid view enables you to measure data without seeing the map. Instead you have up to 34 codes to choose from with a single tap.

Both the Grid view and the Quick Select list selection enable you access feature codes quicker by filtering feature codes by group and category. Groups and categories must be defined in the Feature Definition Manager in the office.

A category is a class of related feature codes, for example, vegetation. For certain measurements or tasks, you might want to group feature codes from different categories into a group for faster access.

With each feature code, different attributes can be stored, which enables you to describe a recorded point or line with more information. Attributes need to be set up in the Feature Definition Manager and cannot be changed or created in the field.

Different properties can be applied to each attribute, for example, if it is optional or compulsory to fill out this attribute, which values are required for this attribute, the permitted length of the text string that you can enter, or available items in a drop-down list.

Photos

Photos can be attached as an attribute using the internal camera of the controller or from photos imported onto the site controller.

The pictures are associated with the measured point and will be available to view in a Trimble Business Center project. Photo attributes are set up in the Feature Definition Manager.

7

Volume and COGO

When collecting data in the field, it is useful to be able to review and edit your data. This section covers reviewing and editing any surface data you have measured, calculating new points and lines in the field, and calculating the volume from measured data.

Review & Edit Data

Use this feature to delete points you may have incorrectly measured. You can also use this feature to calculate volumes of any surfaces that you have measured.

It also enables you to display contours of the surface, which serves as a quick check that you have correctly collected data.

From the COGO menu, tap Review & Edit Data 

.

The Review and Edit Data screen has a list of icons on the left side, which represents all the available functions. These functions can also be accessed directly by pressing the help icon in the upper right of the screen.

Icon	Description
	Compute Volumes
	Create Line/Boundary
	Delete Point/Line
	Generate Measured Contours
	Compute Distance
	Compute Total Distance, including the slope distance between two points
	Compute Area
	Compute Down and Out from Line
	Compute Angle
	Help *NOTE – The help icons are active and the associated task and workflow can be accessed by tapping on the icon.*

To undo an action, tap  . To reset an action, tap  .

Compute Volumes

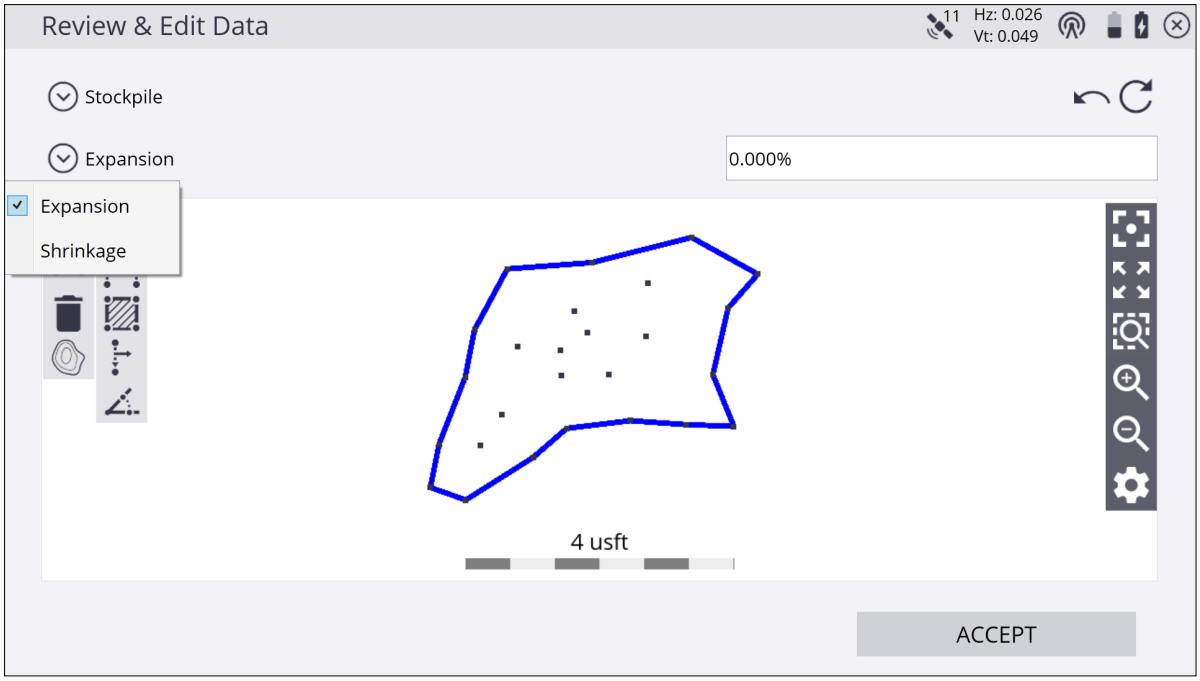
Use the **Compute Volume** option to calculate a volume from the data you have measured. Three types of volumes can be calculated:

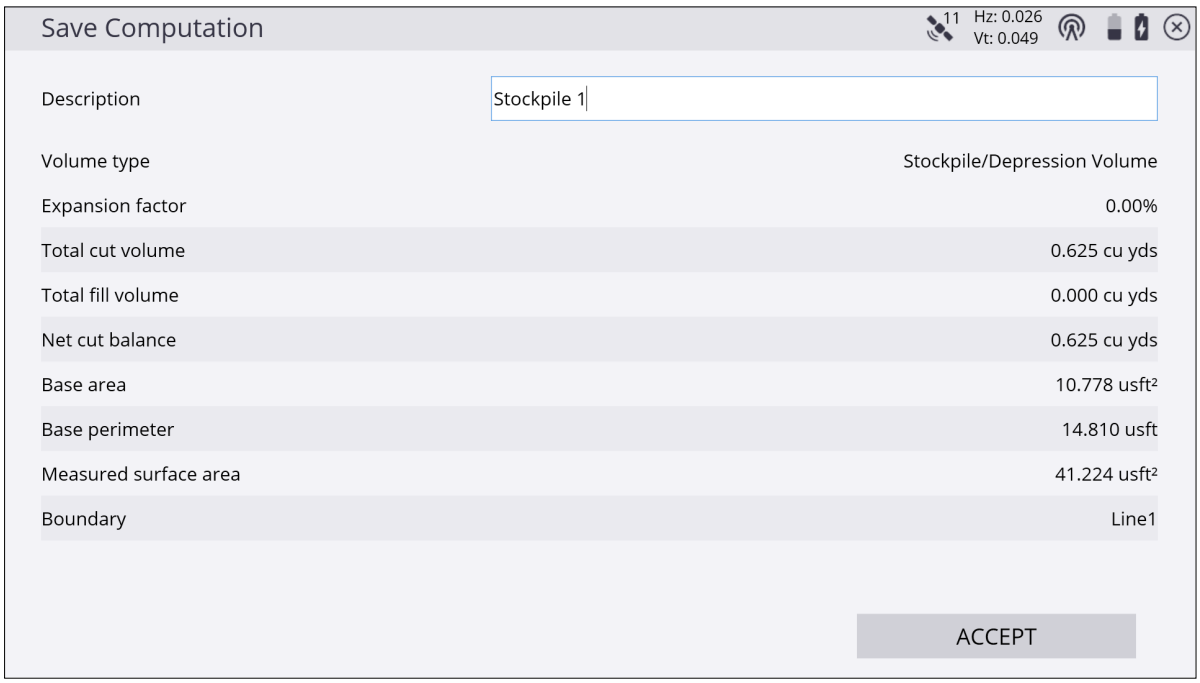
- To a design surface
- To an entered elevation
- To a surface created by the volume boundary (stockpile/excavation volume)

NOTE – You must have measured or created via the appropriate COGO functions a closed volume boundary around the area you wish to measure the volume of.

From the **Review & Edit Data** menu, tap **Contour Measured Surface**  to view contours based on the surface you have measured. This is a useful tool to check for any major errors in measurement that can then be adjusted via the relevant COGO commands. The contours highlight any elevation errors in the data. The software controls the minimum contour interval based on the elevation range of the measured data.

1. In the **Elv. Interval** field, enter a contour interval and then press **Enter**.
2. Tap the **Compute Volume** icon  .
3. Tap on the boundary of the area for which you want to calculate the volume and then tap **OK**.
4. Select the type of volume to compute:
   a. The volume from the surface you have measured to the design surface.
   b. The volume to a user-defined elevation.
   c. The volume of a stockpile/depression – this calculation uses a surface created from the measured line of the selected volume boundary.
5. The following screens show the result of the volume calculation. An expansion or shrinkage factor can be entered to accommodate for material expansion or shrinkage.





The results of the volume calculation are stored in the TaskLog.txt and can be reviewed using the name of the volume in the system log, accessed via Data Management / Log.

Create Points/Arcs

Use the **Create Points/Arcs** option to create design data in the field. You can create new points relative to other points and lines in the work order or in the current loaded design.

From the **COGO** menu, tap **Create Points/Arcs** 

.

A variety of functions are available in the bar on the left. These functions can also be accessed directly by pressing the help icon  in the upper right of the screen.

Icon	Description
	Create a radius point for an arc.
	Create offset points from a line.
	Create an offset point at a certain station.
	Create a mid-point of a line or arc.
	Subdivide a line or arc in segments.
	Create a point at a distance and bearing.
	Enter the coordinates of a stakeout point.
	Create points at the end of a line or arc.
	Free Point Pick.
	Toggle display bar.
	Create a point at the intersection of a line.
	Deflection from Line
	Delete points and lines.

Icon	Description
	Create an arc from three points or two points and radius.
	Create a circle by selecting a center point and either clicking or entering a radius or diameter.
	Create a new line from two points.
	Select points from a list
	Define 2d object elevation
	Access the Help. *NOTE – The help icons are active and the associated task and workflow can be accessed by tapping on the icon.*

Points can be stored as stakeout points or as measured points. A surface can be generated from measured points, which you can then export to the GCS900, AccuGrade, or Earthworks grade control systems for machine guidance through the **Home** / **Import/Export** / **Export to Machine** command.

Key In Roads

This functionality is available to users who have purchased the Roading module.

From the **COGO** menu, tap **Key In Roads** 

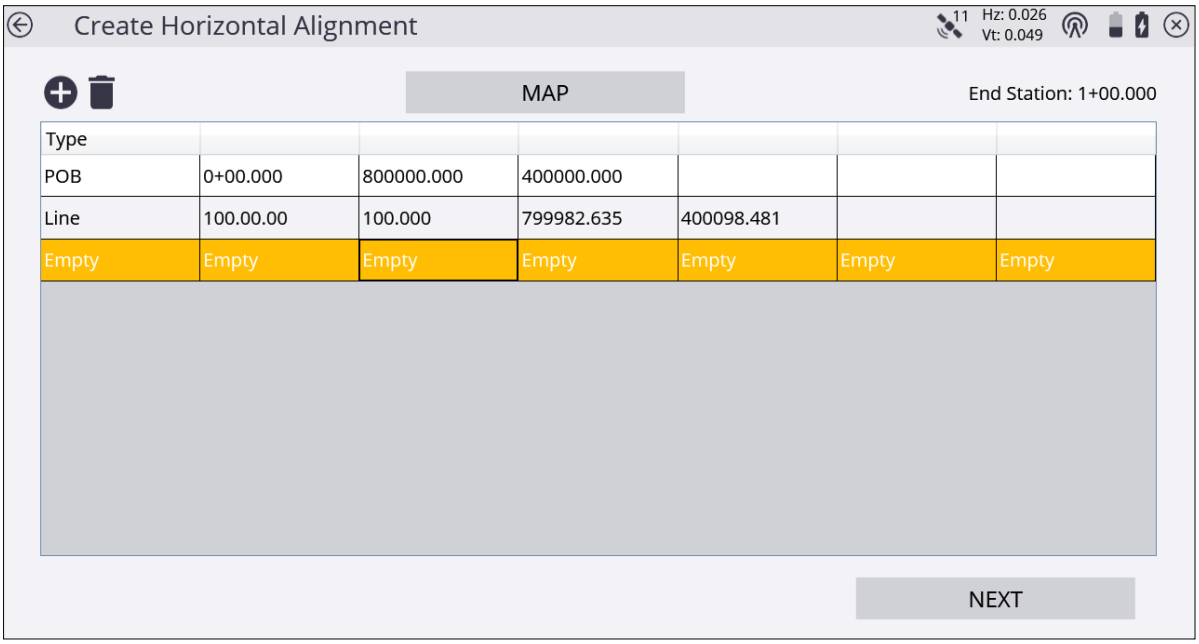
.

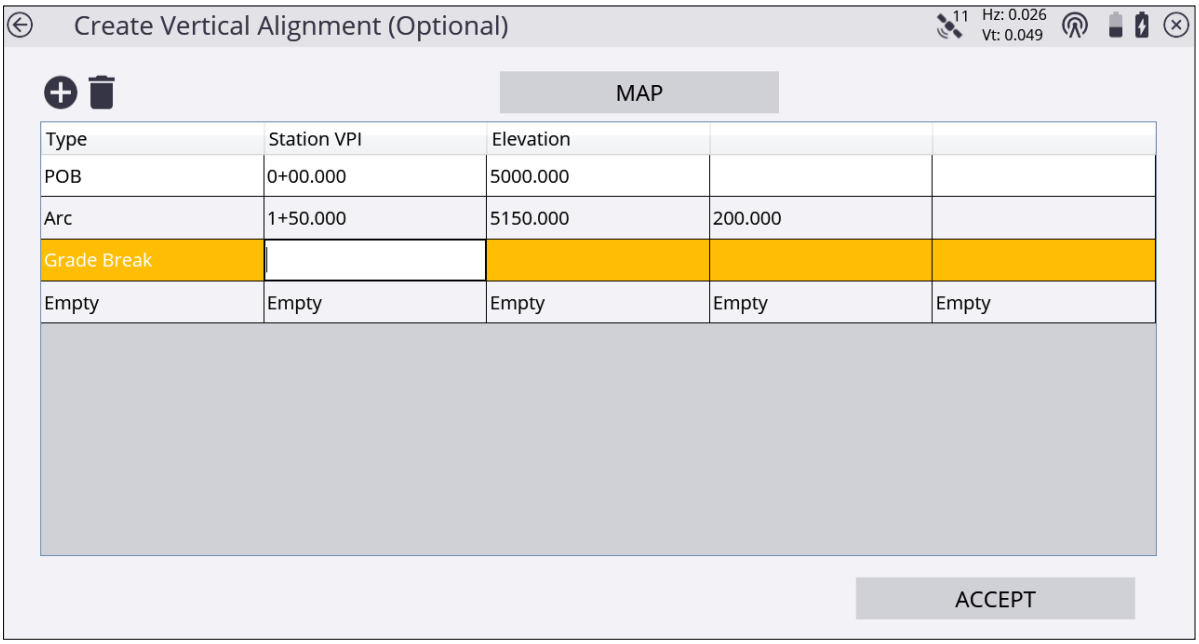
A variety of functions are available in the bar on the left:

Icon	Description
	Create/edit a roadway alignment.
	Create and position road templates.
	Create stakeout points at a certain station and offset from the road alignment.
	Create stakeout points at a certain station and with a deflection angle from the road alignment.
	Change the Key In road entry method.

Creating an alignment

Tap the Create/Edit Roadway alignment icon  . A tabular entry screen appears to enter the geometry for the horizontal and optional vertical alignment.

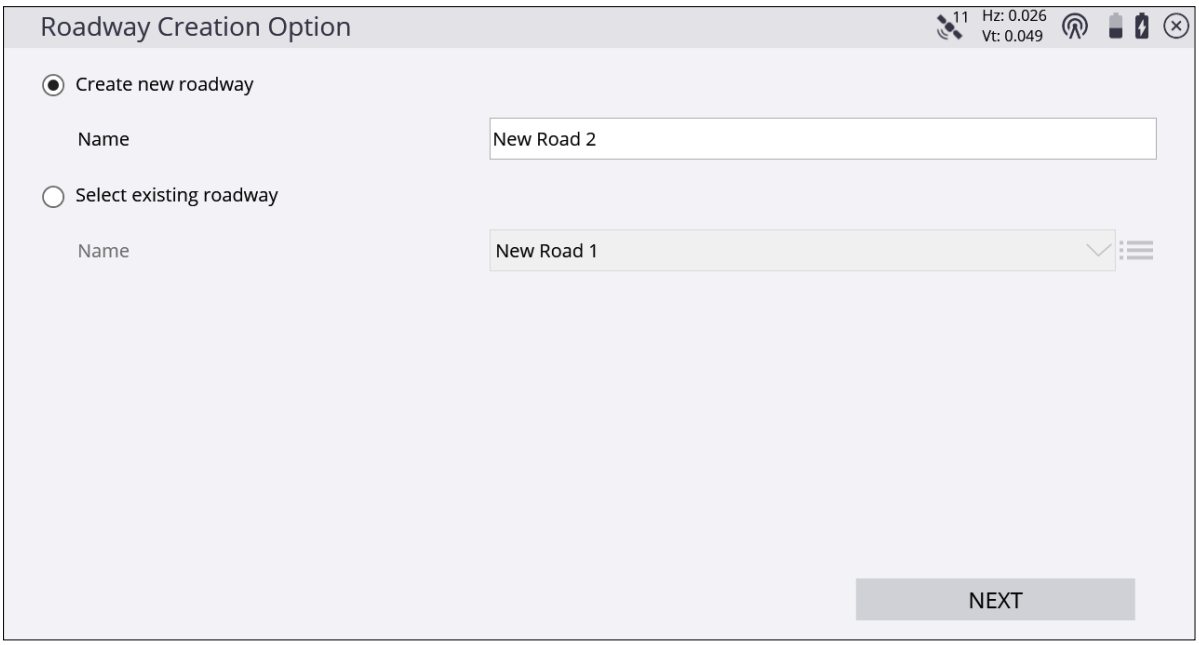




Complete the numeric boxes to define it.

To convert a polyline of the current loaded design or work order in an alignment, tap **Map**.

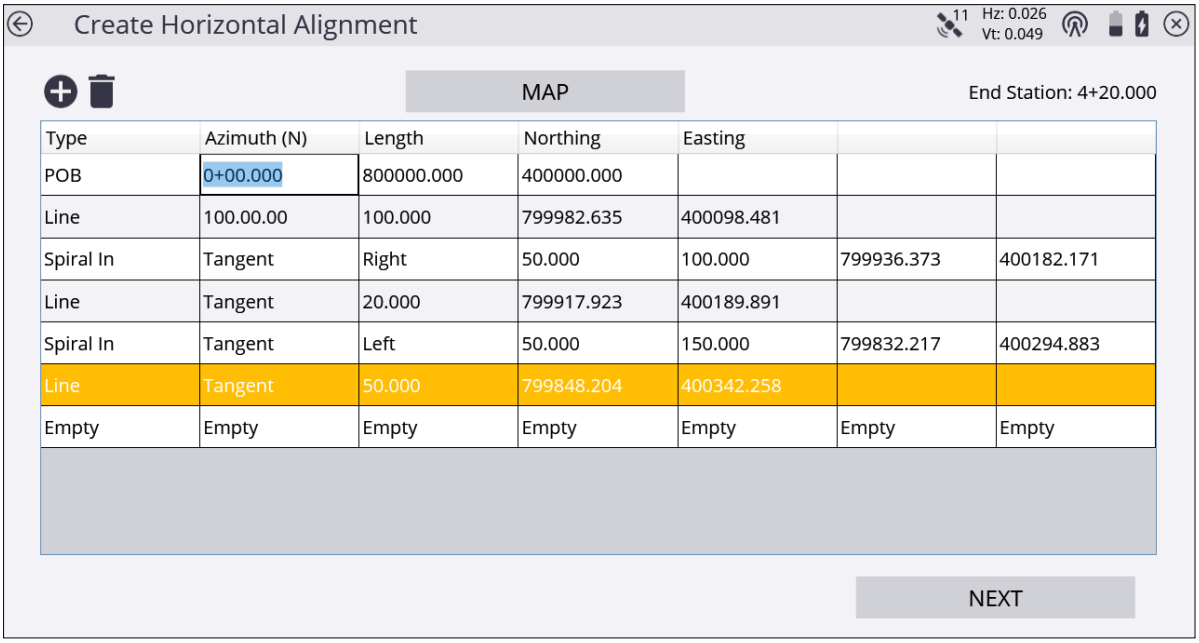
If roads already exist in the current design that were created in the Siteworks software, you are prompted to choose to edit an existing road or create a new one:

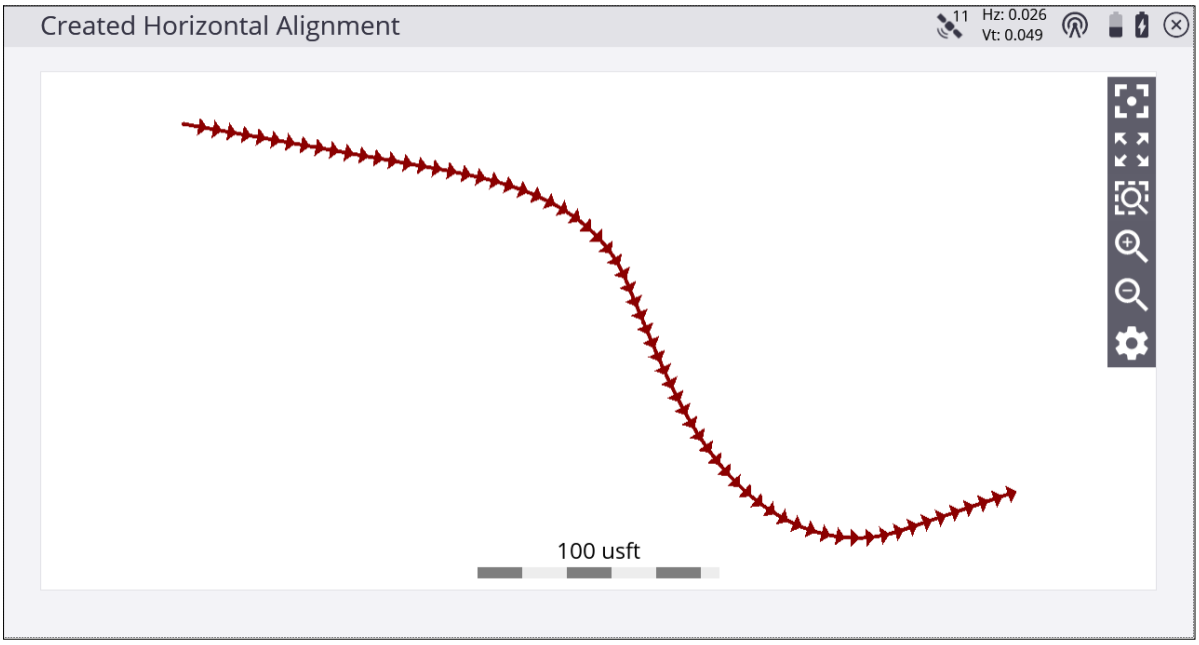


The following table shows the record types that the Siteworks software supports and the data that you must enter for each type. Depending on the Key In Road settings, the alignment can be entered via segments or via coordinates for the points of intersections. The POB is the Point Of Beginning, which is always the first record for a horizontal alignment and contains the start station and coordinates. The azimuth is always automatically computed and appears in the table as Tangent. If you tap on this field, the software displays the actual calculated azimuth. If required, you can overwrite it with your own value. You can also enter the azimuth as a bearing (that is, S 90 W = azimuth of 270°).

Record type	Col1	Col2	Col3	Col4
POB (Point of Beginning)	Station	Northing	Easting	
Line	Azimuth	Length		
Spiral In	Azimuth	Direction	Radius	Length
Arc	Azimuth	Direction	Radius	Length
Spiral Out	Azimuth	Direction	Radius	Length
Combining Spiral	Azimuth	Length		

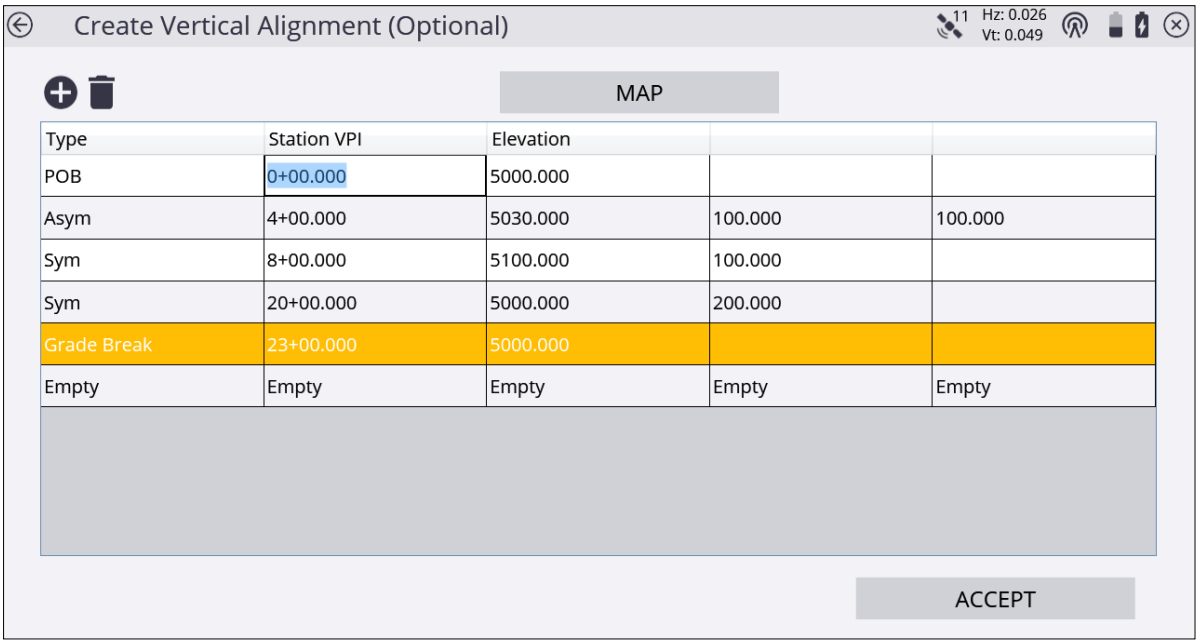
The end station value appears as you enter the road details. Tap Map to see the plan view of the alignment you are creating:





Creating a vertical alignment

Tap **Next**. The **Create Vertical Alignment** screen appears:

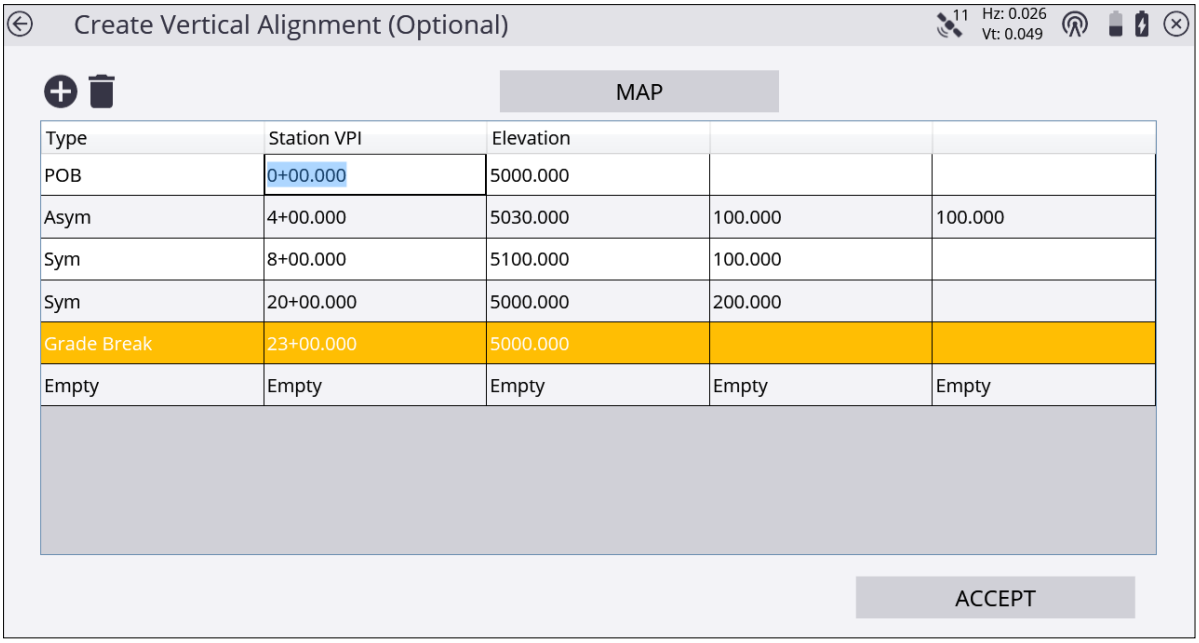


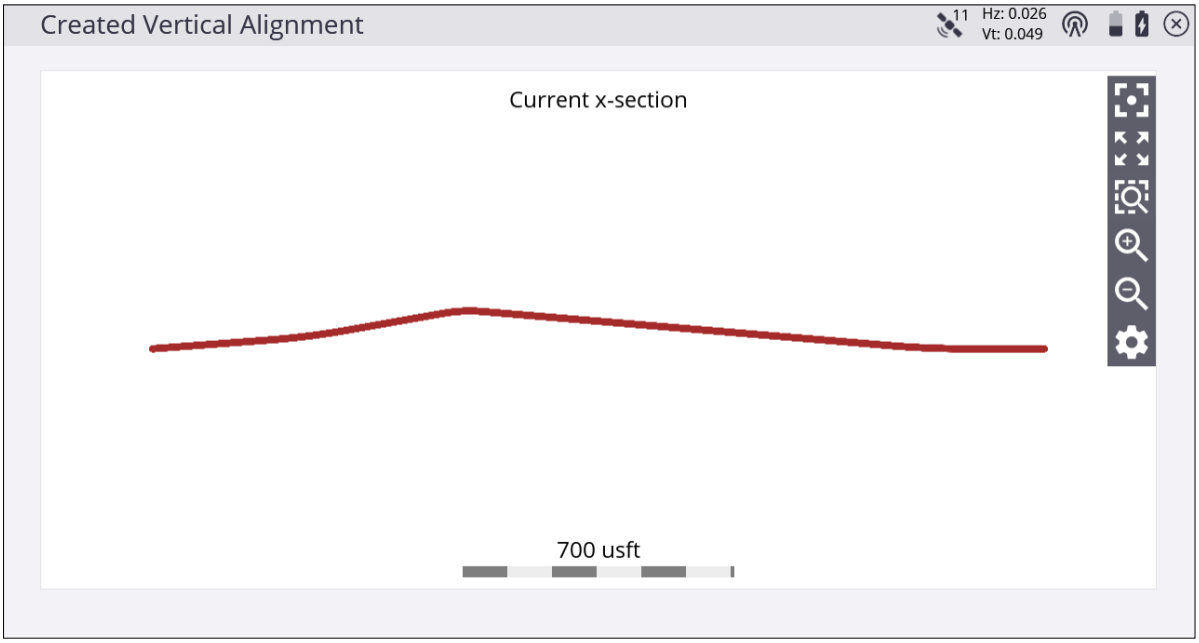
Complete the numeric boxes to define it.

This table shows the record types that the Siteworks software supports and the data that you must enter for each type. The POB is the Point Of Beginning, which is always the first record for a vertical alignment and contains the start station and elevation.

Record type	Col1	Col2	Col3
POB (Point of Beginning)	Station	Elevation	
Arc VPI	Station	Elevation	Radius
Vertical VPI	Station	Elevation	Length
Grade Break	Station	Elevation	

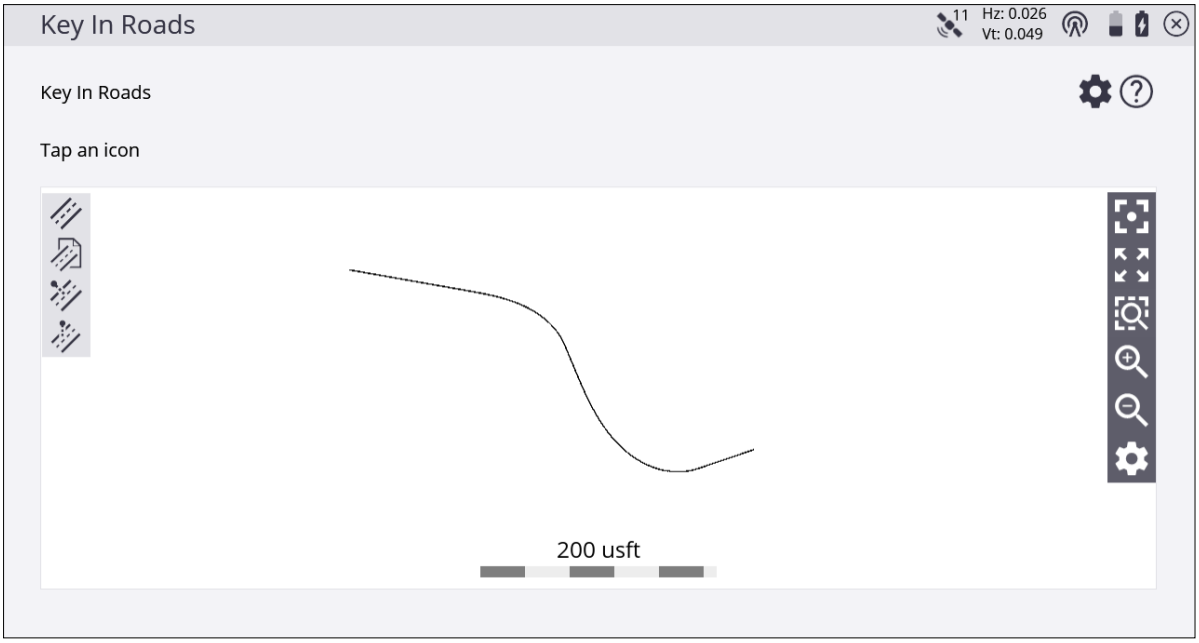
The final station value must be the same as the end station displayed in the previous Horizontal Alignment entry screen. Tap **Map** to see the profile view of the alignment you are creating:





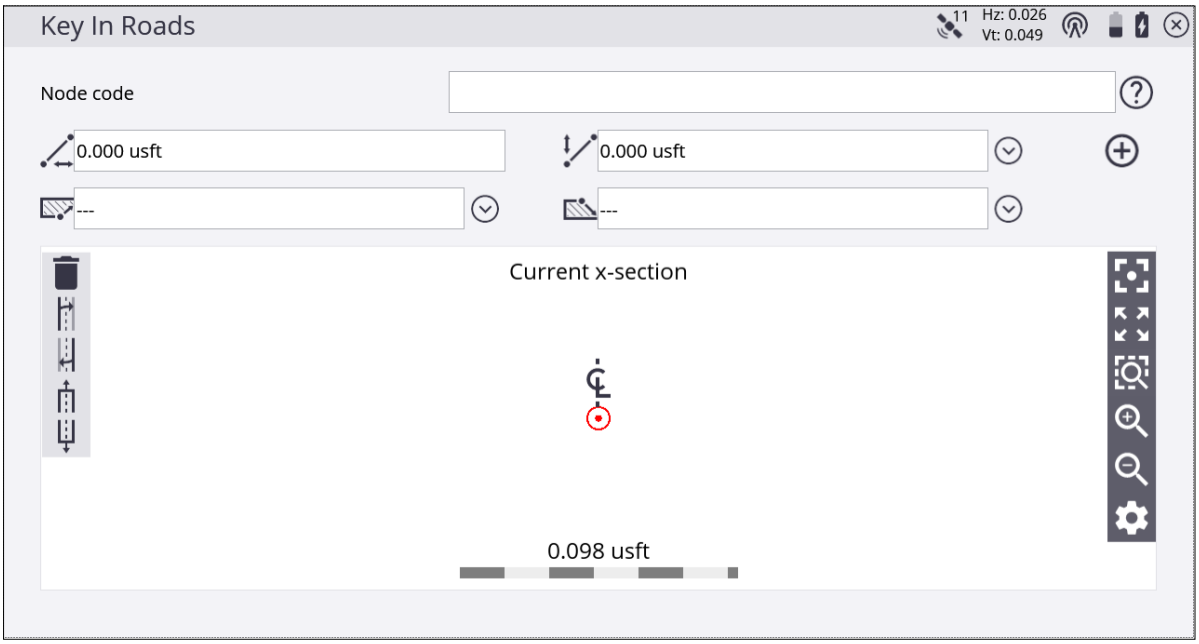
If there is no current design, you are also prompted to create a new Siteworks design.

The main create road screen reappears showing you the plan view of the alignment you have just created:



Positioning and creating templates

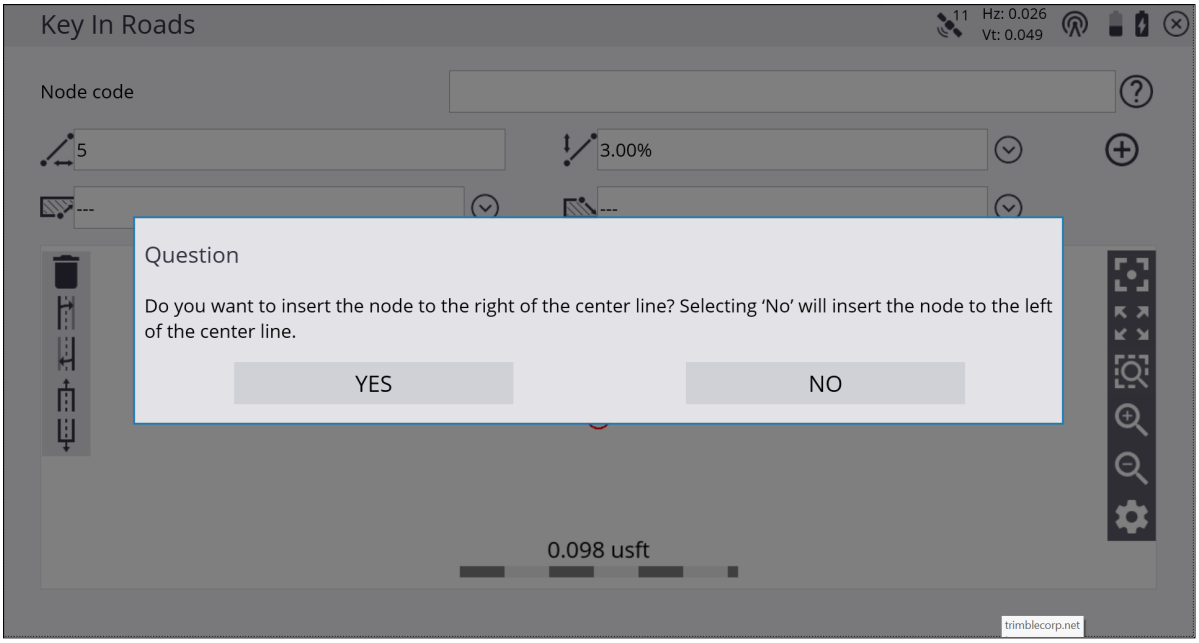
Tap the position and then tap the create road template icon  . The following screen appears:



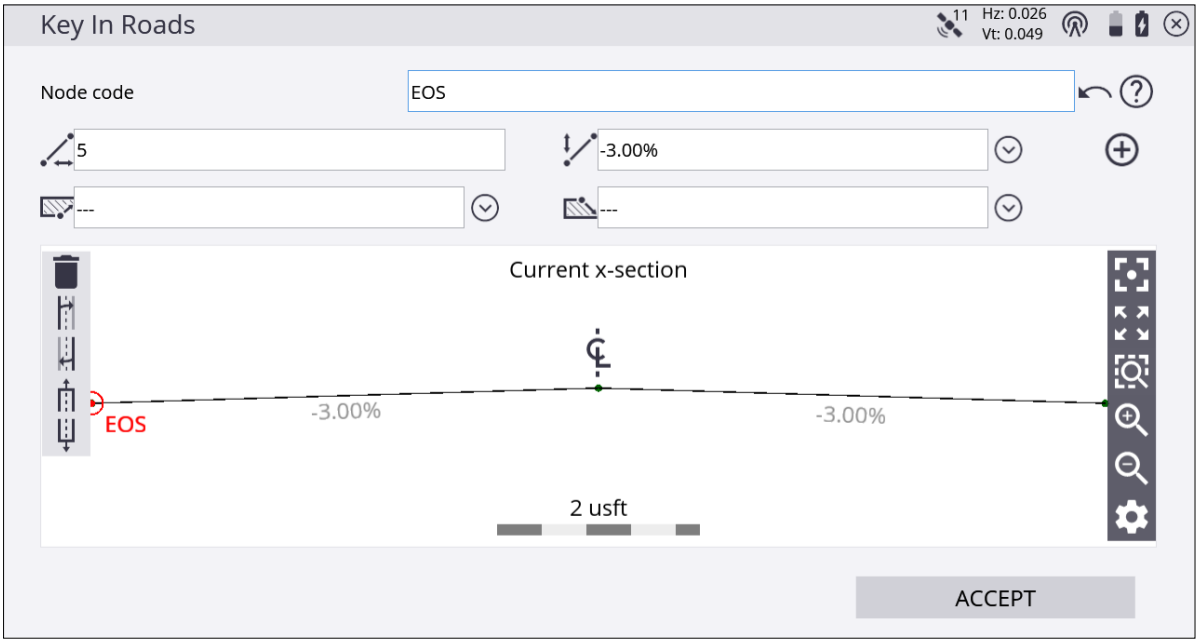
Pick the required station for the template. The following screen appears which allows entering the template.

Button	Description
Node code	Code for the feature node you are about to enter.
	Horizontal distance to the last feature node which was entered.
	Vertical distance to the last feature node which was entered.
	Create Feature Node with the entered values.
	Slope values for the Cut tie.
	Slope values for the Fill tie.
	Delete feature node.
	Copy the left side to the right side.
	Copy the right side to the left side.
	Import a previously entered template.
	Export a previously entered template.

When you first enter this screen, you will be asked if the values apply for the right side or the left side of the road; the values are applied from the centerline.



Tap **Insert** to see the edge of seal on the right side of the road on the cross section.



Copy the right side of the road over to the left side by tapping the copy template button. When the template is complete, tap **Accept**. Enter a name for the template and whether you want to store the template in a library that can be accessed from any site on the controller.

Using templates means you can easily recall them by tapping the import template button. Select a template from the list. The whole definition appears in the cross section view.

You can view the templates either in plan view or cross section view at any station you want. You can also view the templates at stations between definitions. The Siteworks software transitions between the templates.

Creating stakeout points

Two COGO functions are available:

- Create stakeout points at an offset from the alignment. This function can be used for any road, not just ones created in the Siteworks software.
- Create stakeout points at an offset from the alignment at a deflection angle. For example, this can be useful where a drain crosses a road. This feature can be used for any road, not just ones created using the Siteworks software.

8

Stakeout Workflow

The Siteworks software enables stakeout of points, lines, surfaces and roads stored in a design. You can access the **Stake** menu either through the **Home** menu, or by tapping and holding on items in the **Measurement** screen.

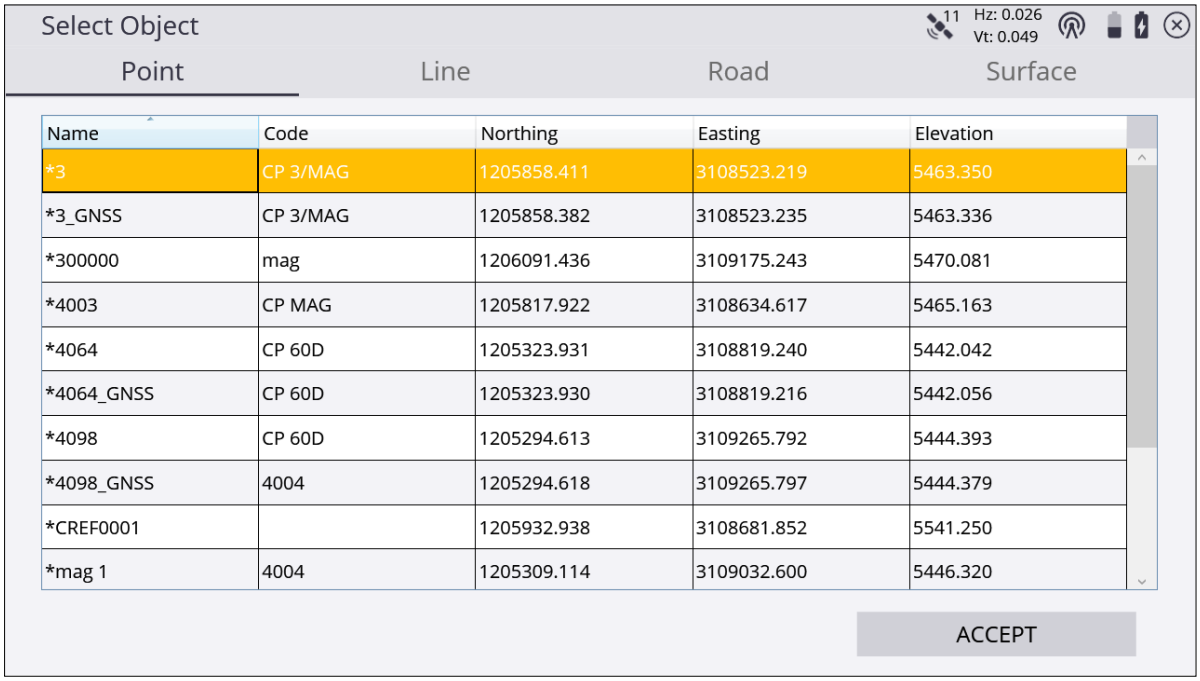
Points

Before you can stake out points, the points must be part of the currently loaded work order. There are a number of ways to get points that were not measured in the field into a work order:

- Enter the coordinates of the point using the Point Manager function.
- Use the Create Stakeout Points COGO functions.
- Import stakeout points during the project creation process.
- Import a point file through the **Home** / **Data Management** / **Point manager** / **Import Point File**.

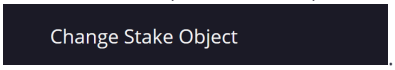
1. From the **Measurement** screen, tap the **Home** button and then tap **Stake**.
2. Select a point using the list at the top right of the screen and then select a stakeout point in the **Points** tab. Alternatively, select the point directly from the map (tap and hold and then select **Stake Point** from the pop-up menu). If there is more than one object available in this area, a list of different objects appears that you can select from.

   The list puts different stakeout items into separate tabs; Point, Line, Road or Surface:

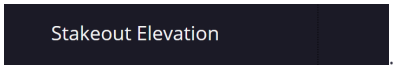


3. If you need to calculate a stakeout point from design data, tap **Home** and use the functions in the **COGO** menu.
4. Multiple stake methods (point, side slope, and catch point) are available. For information about slope staking, see Slope staking, page 108.

5. Use the values in the info bar (for example, GO) to navigate to the point. A small green arrow between your current position and the stakeout point provides you with guidance. In addition, a large arrow on the top right of the screen will point in the correct walking direction to the point after the software recognizes in which direction you are currently moving. When using a map rotation in travel direction or following a selected alignment, an additional North Arrow on the top left indicates North so that the values in the info bar can be used to navigate to the point.
6. When you are close to the stakeout point, the software switches into the Fine Stake mode. Additional guidance arrows appear on the top right corner of the map to indicate the remaining distance in each direction.
   - When staking with a GNSS, the fine navigation arrows will be displayed in a north "up" orientation or the direction of approach when the fine navigation arrows appear, depending on the setting of the **Stakeout Guide** in the **Design** tab of the **Map Options** screen.
   - For staking performed with a total station, the fine navigation arrows will be oriented depending on the connection method to the total station.
   - For Bluetooth and cable connections, the directions will be as if you were standing behind the total station looking towards the point.
   - For radio connections, the directions will be as if you were standing at the prism pole looking towards the total station.
7. Once you are within horizontal tolerance (set via the **Home Menu** / **Settings** / **Stakeout**), the dot in the circle of the stake guidance turns yellow. After tapping the **Measures** button  , a stakeout report for this point appears. The software remembers which tab of the stakeout report was last viewed and opens the same tab after staking the next point.
8. Select the **Edit stakeout name** check box to store the staked point using a different point name and point code.

   In another tab, a graphical diagram shows how to put an elevation mark on the stake. The software does all the calculations for you. The way that the software calculates the elevation mark and cut/fill depends on the stakeout settings in the Stake Marking Method settings in the **Home Menu** / **Settings** / **Stakeout**.
9. After the stakeout of the point has been completed, the software returns to the **Stakeout Selection** screen. If during the stakeout process a different point is required, tap the **Home** menu and then tap **Change Stake Object** 

.

10. To change the stakeout elevation to a different value, tap the **Home** menu and then tap **Stakeout Elevation** 

.

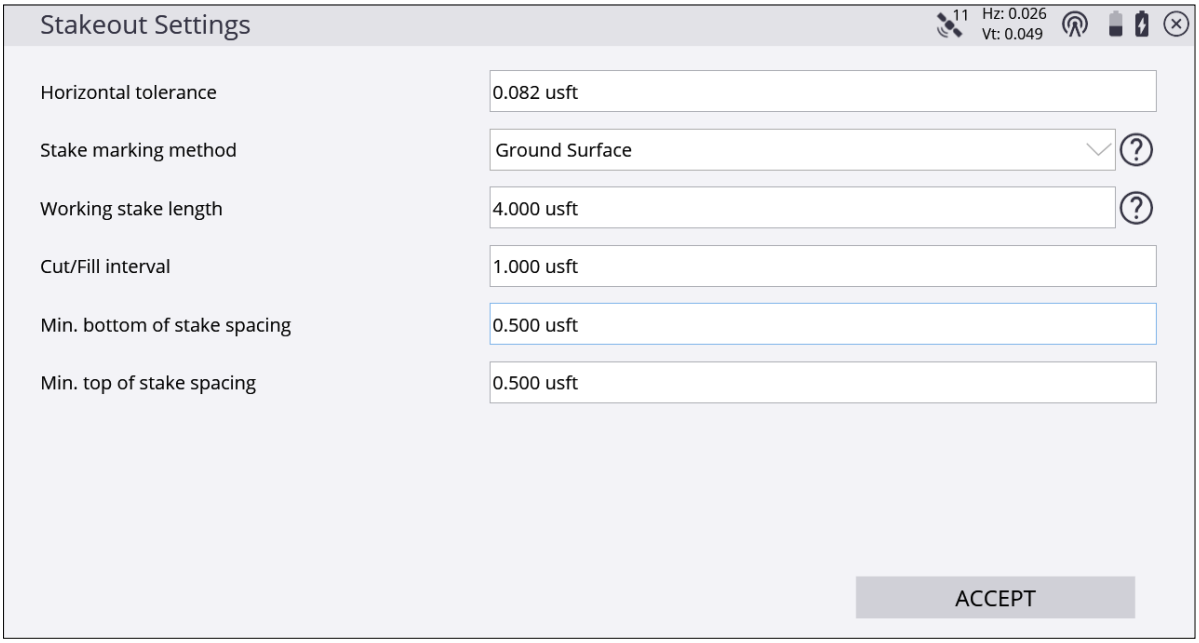
Stakeout Settings

To access these settings, from the map screen, select the menu and then tap **Settings** / **Stakeout**. The software supports three elevation reference methods used to mark cut depths or fill heights on a grade stake or location/grade stake:

- Measuring the cut/fill reference mark from the ground surface
- Measuring the cut/fill reference mark from the top of the stake
- Cut/fill reference from the measured point

These methods help you to establish a cut/fill reference mark on the stake at a specified cut/fill measurement interval, such as at one-foot increments. If you choose to place a cut/fill reference mark on the grade stake, a Stake Marking report helps you establish the position of the reference mark on the stake and helps you correctly label it.

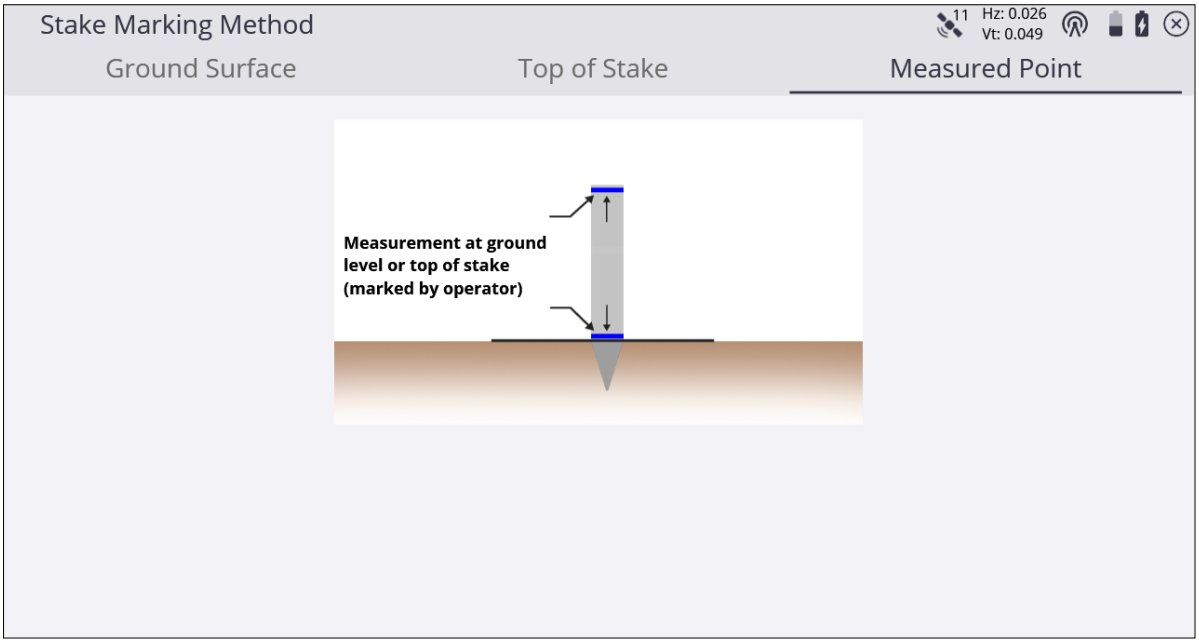
Typically, you will use one of the above methods consistently. Usually, you use only one method. When you first receive the software, switch to the correct setting. The software then uses that setting for all stakeout operations. When you tap **Measure** during a stakeout operation, the software converts the measured elevation, design elevation, and computed cut depth or fill height into information that you can then write on the stake. It also informs you where to mark the stake based on the settings that you enter in this dialog:



The stakeout tolerance is also entered in this dialog.

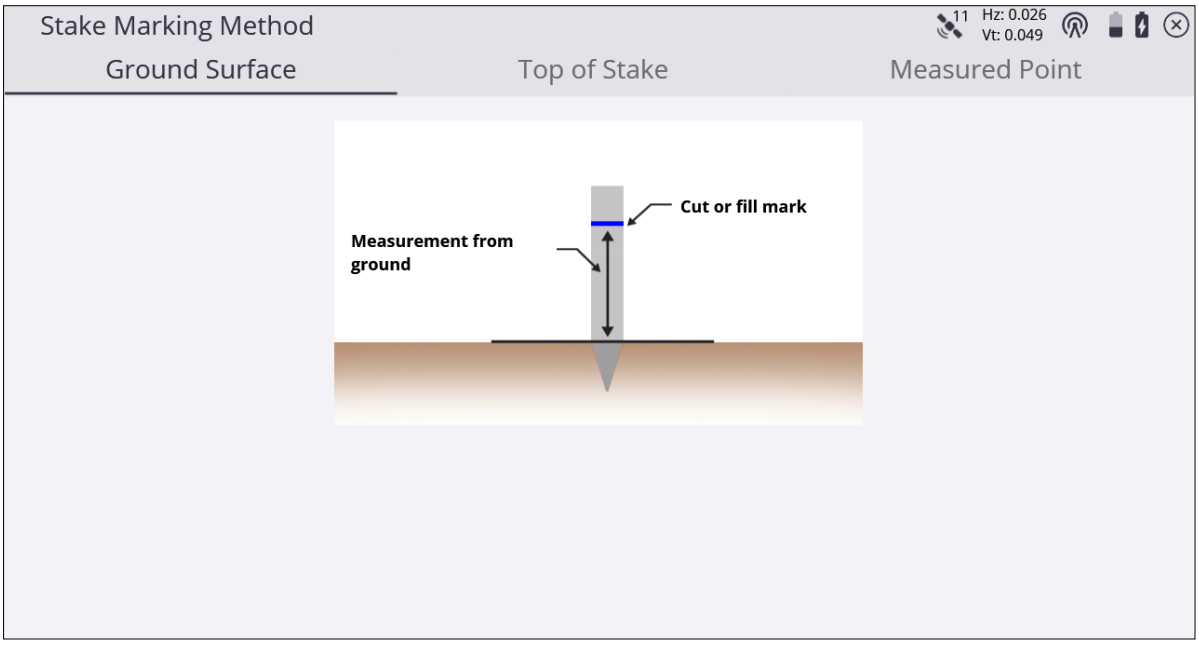
Measured Point

This method enables you to label a grade stake with the required cut depth or fill height as measured from the measured point, which can be either the top of stake or the current ground surface. If you choose to mark the stake with the cut depth or fill height as referenced to the measured point, the software simply informs you of the direct cut or fill measurement from that point. In this case, you can mark that measurement on the stake, using your normal convention to indicate from where the measurement is referenced.



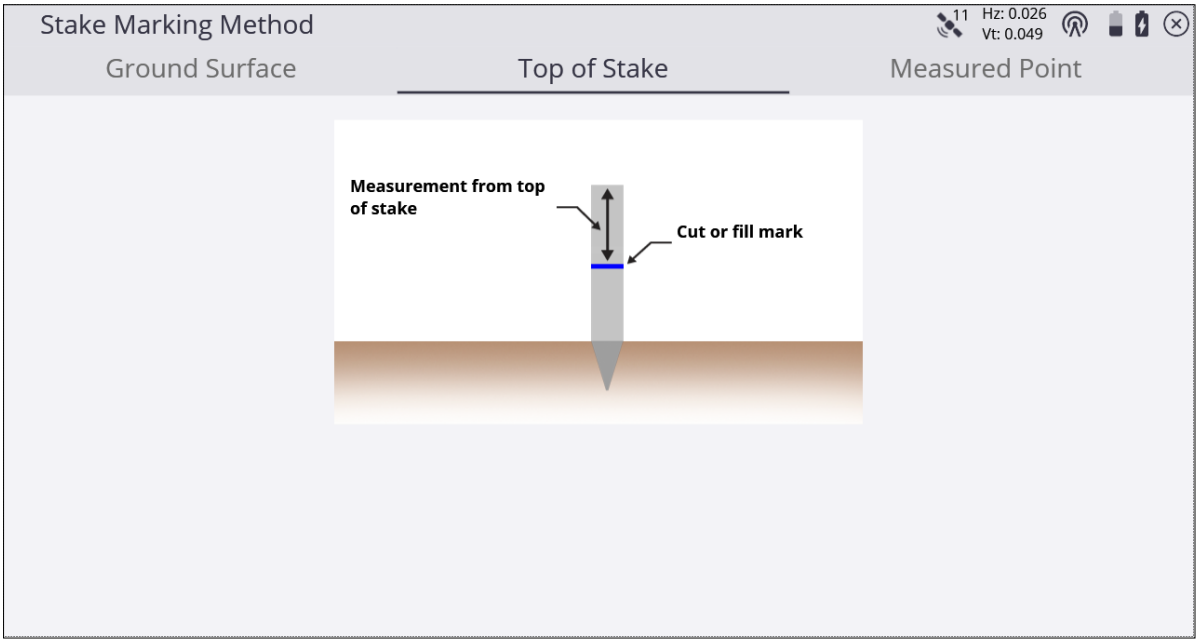
Ground Surface

With this method, the software guides you to the horizontal location of the stakeout point. The software generates a Stake Marking report that shows the distance from the ground up to where you must mark the stake. It also shows the value of cut or fill to mark on the stake.



Top of Stake

With this method, the software guides you to the horizontal location of the stakeout point. You then hammer the stake into the ground and measure the top of the stake. You can change the antenna height for this measurement in case you want to take the receiver off the pole and place it directly on the stake. The software shows the distance from the top of the stake down to where you must mark the stake. It also shows the value of cut or fill to mark on the stake.



Lines

The lines you want to stake out must be part of a design map in the currently loaded design. There are multiple ways to get lines into a design, in addition to measured lines:

- From a DXF foreground map with lines created in the Business Center - HCE Software.
- By creating lines from points using the COGO functions

1. From the **Measurement** screen, tap the **Home** button and then tap **Stake**.
2. Select a line using the list at the top right of the screen and then select a stakeout line in the **Line** tab. Alternatively, to select a line directly from the map, tap and hold on it. If there is more than one object available in this area, a list of different objects appear where you can specify your selection. When using the Business Center - HCE software, names can be assigned to lines, which improves the orientation.

You can also create a new line from points in your loaded design or work order by tapping the New line icon in the top right corner of the screen:

3. If necessary, before confirming the selection, change the line direction using the button on the top right of the screen. This function is not available for lines selected by tap and hold.
4. Different stake methods (side slope and catch point) are available. See page 108.
5. Enter the station to be staked out or tap on the line where you want to stake it and then tap ACCEPT. To change the settings related to the line staking, such as desired horizontal and vertical offsets, stakeout elevations, station advancement interval and if corner/tangent points should be created, tap .
6. The map view then guides you to the point. To help you find the point, the guide arrow in the upper right side needs to point up the screen to show that you are traveling directly toward the point. You can turn the guide arrow off and on in the Design tab of the Map Options dialog, accessed by tapping . The information bars at the top of the screen can be customized using the Info Bar/Panel option. The display shows the design elevation for the point, the amount of cut or fill required to get to that elevation, and how far and in what direction you need to travel to get to the point.
7. The default map view has the direction north pointing up. You can change this so that the guide arrow is pointing in the direction you are walking by changing the map

rotation in the Map Options dialog. It is recommended to turn on the cut/fill lightbar in the top or bottom panel on the left to graphically show the cut and fill.

8. When you are close to the selected line point, the software switches into the Fine Stake mode. Additional guidance arrows appear on the top right corner of the main panel to indicate the remaining distance in each direction. The screen is oriented to the last moving direction before the Fine Stake mode was activated if map rotation in travel direction was selected.
   - When staking with a GNSS, the fine navigation arrows will be displayed in a north "up" orientation or in the direction of approach at the time the fine navigation arrows appear. Set this Stakeout guide option in the Design tab of the Map Options screen.
   - For staking performed with a total station, the fine navigation arrows will be oriented depending on the connection method to the total station. The direction of the fine stakeout guide will be in the direction of approach when the fine navigation arrows appear if Approach Orientation is selected. If North/Instrument Orientation is selected, the behavior will be as outlined below.
   - For Bluetooth and cable connections, the directions will be as if you were standing behind the total station looking towards the point.
   - For radio connections, the directions will be as if you were standing at the prism pole looking towards the total station.
9. After tapping Measure, a stakeout report appears. The software creates a Stake Marker report. A graphical diagram shows how to put an elevation mark on the stake. The software does all the calculations for you. The way the software calculates the elevation mark and cut/fill depends on the stakeout settings in the Trimble icon menu. The software remembers which tab of the stakeout report was last viewed and opens the same tab after staking the next point.
10. Instead of staking a certain station it is also possible to stake a line at random stations using the buttons on the status bar on the right side of the main panel:

Tap...	to...
	stake at fixed station intervals starting at a certain station.
	stake at random and arbitrary station intervals anywhere along the line, based on your current station location perpendicular to the line. *NOTE – The fine stakeout guide arrows will not appear when in random mode.*

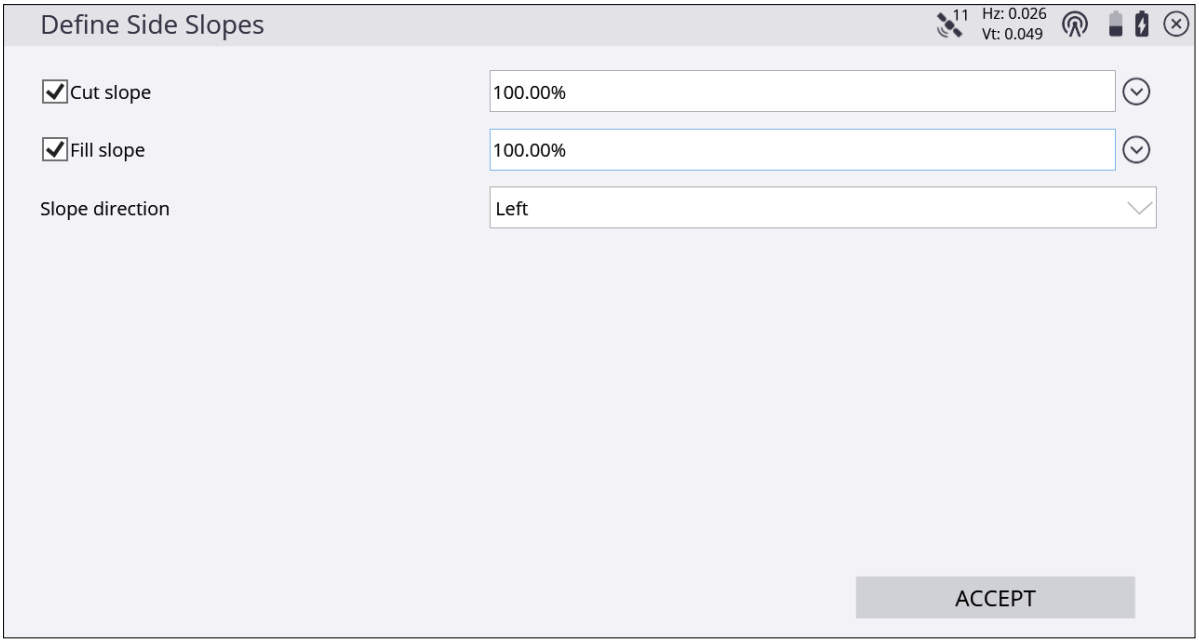
Settings available for staking lines include options to enter a horizontal and vertical offset for the stakeout line. The line elevation can be defined using different methods and the start station, station interval (increment distance), and whether or not to automatically advance to the next station can also be determined. If a horizontal offset is applied and yes is selected in the **Create tangent/corner points** selection menu, three stakeout points are created at each corner to help you stake out the lines. Access these settings from the Home menu under **Stake/ Line settings** or by tapping the Line Settings button  on the **Select Line Station** selection screen or in the top left of the main panel.

Slope staking

For both stakeout point and stakeout lines, side slope and catch point stakeout modes are available that enable staking out the slope or the catch point between this slope and the existing ground. This function can be applied to any earthwork operation that involves a tie to the current ground surface. Examples include staking earthworks for pad placement, earth dams, site drainage, ponds, lagoons, embankments, and keyways.

The tie-slope can be projected from a 3D point using a bearing or from a 3D Line. After defining either one of those, the stakeout process is very much alike. Tap on the stake mode icon on the bottom of the status panel on the right side of the main panel:

You can then select line, side slope or catch point. Use the **Define Side Slope** dialog to select the direction of the slope and whether you define a cut and/or fill side slope to the left or right of the reference line. To select what the slope designation is to be based on, alter the selection in the **Slope direction** field.



The software computes the elevation of the reference line at that point, and projects the designated cut and/or fill side slope magnitudes from that reference point, through your position, along a line referred to as the slope indicator. The slope indicator line is shown in the map view, extending from the reference line to the currently predicted catch point location. In predicting the location of the catch point, if you have defined both cut and fill slope magnitudes, the software determines whether the cut slope or fill slope is applicable at the reference point.

When staking building pad side slopes, the building pad has both internal and external right-angled corners. If you are staking external corner points, then the software automatically calculates the side slope as projected radially from the corner point. At an internal corner, the software calculates the catch point at a bi-sectional angle.

The Catch point option enables you to stake out points where the side slope intersects the ground surface as it is found to exist. As you stake catch points at fixed intervals, you should adapt to the existing terrain and be aware of the effect that it may have on the location of the daylight line. You can freely switch between the Stake at fixed intervals mode and Stake at randomly chosen interval modes.

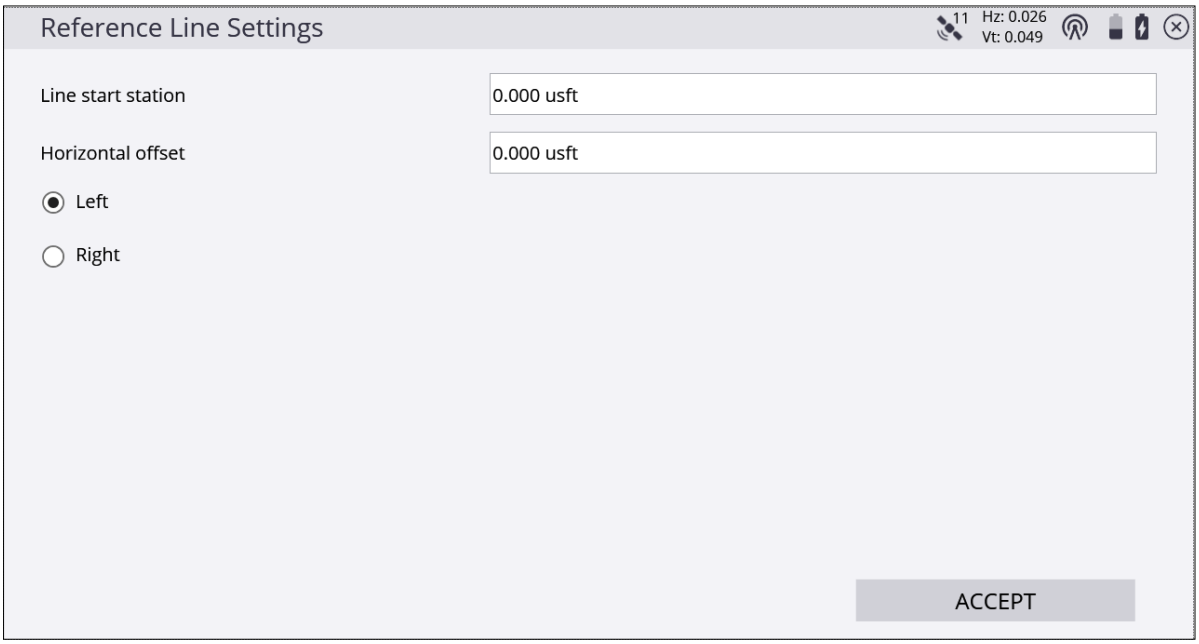
The Line option enables you to stake out the side slope's reference line. The software guides you to the line at the nearest point to where you currently are, or to a specific station. The Side Slope option enables you to place a grade stake at any required location on the side slope between the reference line and the catch point. You can toggle between the different stakeout methods by selecting the icon at the bottom of the status bar on the right side of the main panel.:

Icon	Description
	Stake catch point
	Stake side slope
	Stake line

Reference Line

While staking out different objects like a point, line, surface or road, a separate line or alignment can be selected and referenced to record the station and offset values relative to the reference line in the stake record. To select a reference line or alignment, go the Main Menu / Settings and then tap Reference Line. Select the reference line from the map. Alternatively, tap and hold on a line you want to select as the reference line, and choose Select as Reference Line from the pop-up menu.

When a line is selected to be used as a reference line, an optional horizontal offset and a line start station can be applied.



The station and offset to this reference line can be displayed in the info bar as reference station (R. Sta) and reference offset (R. Off) while staking the actual object. To deselect the reference line, return to the Reference Line in the main menu / Settings and then click on a blank area of the map screen. Alternatively, tap and hold on the original line (not the reference line) and choose Unselect Reference Line.

Surfaces

Use the **Stake Surface** feature to put grade stakes over a design surface indicating the cut/fill to it. The surface you want to stake out must be in the currently loaded design. There are multiple ways to get a surface into a design:

- From an imported TTM surface file (this can be created in the Business Center - HCE software).
- By creating a surface design from an existing Siteworks work order measured surface points and lines, using the **Data Management** / **Surface as Design** feature.

1. From the **Measurement** screen, tap the **Home** menu and then tap **Stake**.
2. Select the surface using the list icon  on the top right of the screen and then in the **Surface** tab select a stakeout surface.
3. Use the values in the display bar to indicate the current cut/fill of the surface.
4. After tapping **Measure**, the software creates a Stake marker report. A graphical diagram shows how to put an elevation mark on the stake. The method that the software uses to calculate the elevation mark and cut/fill depends on the Stakeout Settings in the / **Settings** / **Stakeout**.

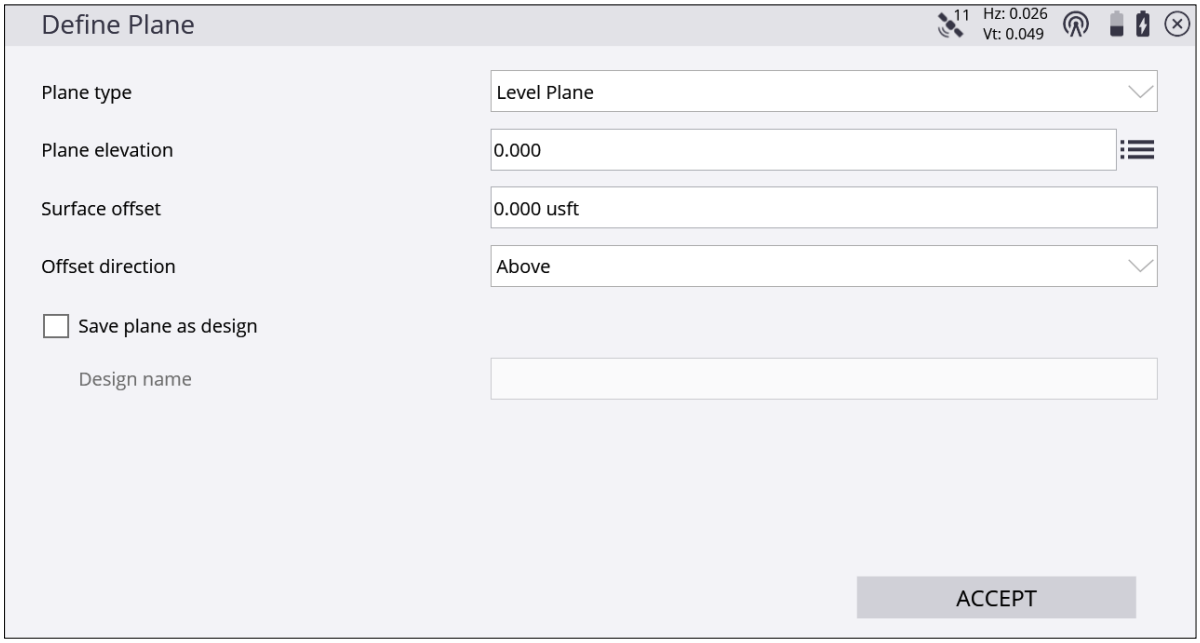
Planes

Use the **Stake Plane** feature to define a plane similar to how a grade laser works and then use a GNSS or a Total Station to get an indication of the resulting cut/fill between the current position and the plane. There are three options to create a plane to stakeout:

- Level Plane – Choose an elevation to define the level plane.
- Sloping Plane – Choose an origin point and elevation, a main slope, and an optional cross slope.
- Three Point Plane – Create a plane from three points.

1. From the **Home** menu, tap **Stake**. This command is also accessible in the main panel by clicking on the icon shown below.

2. On the top right corner of the **Stake Object** screen, tap the **Define Plane** icon . The **Define Plane** dialog appears:



3. Select and configure the plane you want to stake.

   Level plane is a flat surface with a defined elevation; sloping plane is defined by an origin point, main slope, and cross-slopes; three-point plane is defined by picking or measuring three points.

4. To save the specified plane as a design, select the **Save Plane as Design** check box and enter a name into the **Name** field. The plane will then be available in the site's Design folder and available for loading through the **Project** menu or loaded as a second

surface.

5. Use the values in the display bar to show the current cut/fill of the plane.
6. To record a point and view the stakeout report, tap the **Measure** icon in the main panel.

Roads

The road or alignment that you want to stake out must be part of the currently loaded design. These files are saved as .PRO files. There are multiple ways to get a road into a design:

- From a road corridor created in the Business Center – HCE software and exported through the **Field Data** menu.
- Convert LandXML files with the SCS Data Manager.
- Export a road from the Terramodel® software.

If you have the Road module installed and a Road design loaded, you can select a road/alignment from the map by tapping and holding on it, or from the list in the **Stake** menu and then selecting one of the three road staking methods that are available:

- Roadway feature staking(see Roadway feature staking, page 115)
- Catch point staking(see Catch point staking, page 122)
- Location on Surface(see Location on Surface, page 125)

If no Roading option is installed on the controller, a warning appears.

Roadway segment staking

The grade stake marking preferences in the **Stakeout Settings** dialog also apply to staking roadways. Staking methods for catch point staking can be determined in the / **Settings** under **Catch Point Stake Settings**.

1. From the **Measurement** screen, tap the **Home**menu and then tap **Stake**.
2. Select a road or alignment using the list at the top right and then selecting an alignment in the **Road** tab. Alternatively, select an alignment directly from the map by tapping and holding. If there is more than one object available in this area, a list of different objects appears where you can specify the selection.
3. Enter a station or select one from the map at which to stake out the road feature. To change the settings related to the road staking, such as the desired subgrade offset, station advancement interval, what tangent points to automatically stop at, guide line offset, and the view settings, tap

 .

After selecting the station, a cross section of the roadway appears. It notes the location of each roadway feature as a node on that cross section. The Select Roadway Feature screen appears.

4. Select the roadway feature node that represents the roadway feature that you want to stake out. To select a node, do one of the following:
   - Select from a node list (tap the list icon in the top right corner).
   - Tap the required node.

   If the node is on the finished grade model, the name of the node would be SHLD, for example. If the node being selected is on the subgrade adjusted surface, it would be named SHLD-0.250; meaning it is the SHLD node, but it has a -0.250 m offset. To enter a subgrade offset, either tap  or access the road settings by selecting Settings / Roads.

   If you are navigating up station, the normal cross-section view is displayed left to right as expected. If however, you are navigating down station, then the cross section would normally be back to front. To reverse the view of the section, change the view setting to up station or down station entered by either tapping  or accessing the Road Settings by selecting Settings / Roads.
5. Let the software guide you to the point to be staked on the selected feature using the values in the info bar. To help you find the point, the guide arrow needs to point up the screen to show that you are traveling directly toward the point. You can turn the guide arrow off and on in the Design tab of the Map Options screen. The information bars at the top of the screen can be customized using the Customize info bar option in Settings / Info Bar/Panel. By default, the info bar shows the design elevation for the point, the amount of cut or fill required to get to that elevation, and how far and in what direction you need to travel to get to the point. You can scroll through the different values using the grey arrows on either side of the info bar or "flick" the menu. The default map view has the direction north pointing up. You can change this so that the direction you are walking is pointing at you by changing the map rotation the Rotate tab of the Map Options screen. It is recommended to turn on the cut/fill lightbar and the Cross Section in the top or the bottom panel on the left of the main panel. The lightbar will graphically show the cut and fill.
6. When you are close to the required line point, the software will switch to the Fine Stake mode. The design map will be overlaid by guidance arrows to indicate the distance in

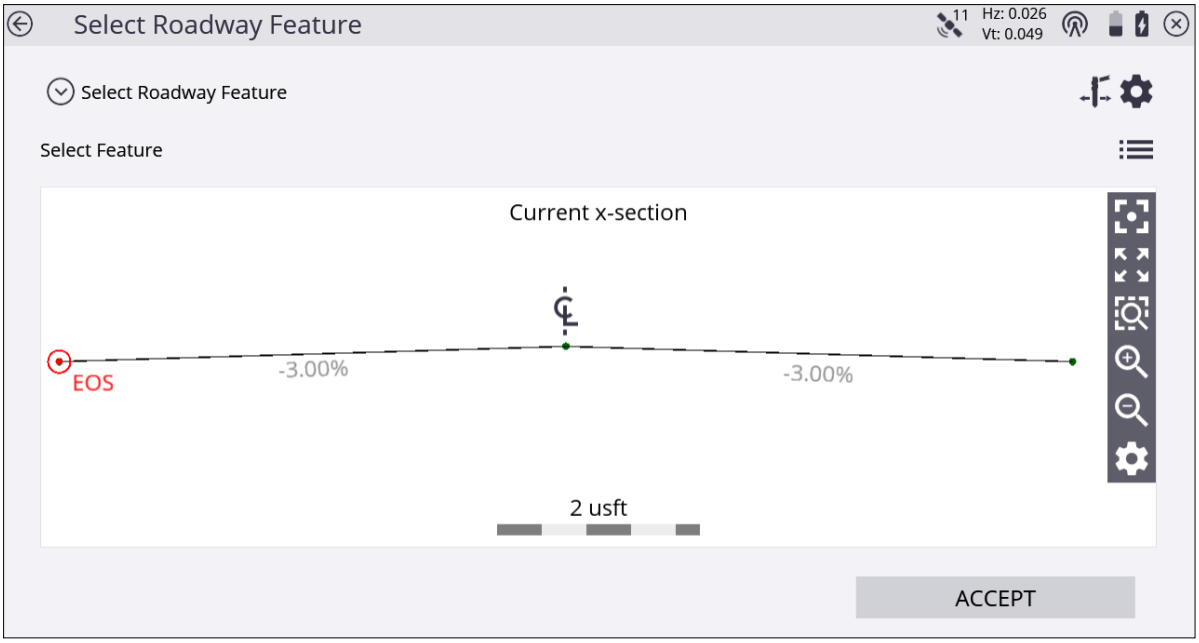
each direction. The screen is orientated according to the map rotation settings in the map options function.

Once you are within tolerance, the center of the guidance arrows will become solid yellow. After tapping **Measure** a stakeout report appears, indicating the storyboard items that can be written on the stake. A graphical diagram indicates how to put an elevation mark on the stake. The method that the software uses to calculate the elevation mark and cut/fill depends on the **Stakeout Settings** in the / **Settings**/ **Stakeout**.

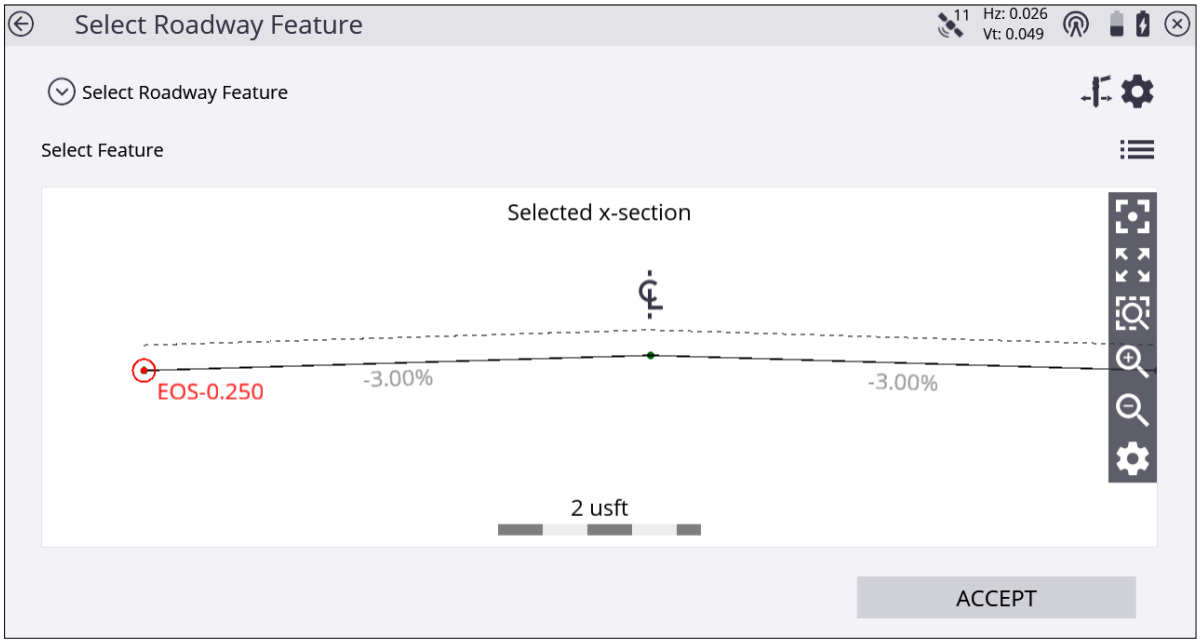
7. Instead of staking a certain station, you can also stake an alignment at random stations based on the location perpendicular to the centerline of the alignment, using the buttons on the bottom right in the status bar.

Roadway feature staking

1. From the **Measurement** screen, tap the **Home menu** / **Stake**.
2. Select a road or alignment using the list at the top right of the screen and then select an alignment in the **Road** tab. In the **Stake Method** drop-down menu select **Feature** and click **ACCEPT**.
3. Enter a station or select one from the map at which to stake out the road feature. The **Select Roadway Feature** screen appears. A cross section of the roadway notes the location of each roadway feature as a node on that cross section.
4. Select the roadway feature node that represents the roadway feature that you want to stake out. To select a node, do one of the following:
   - Select from a node list (tap in the top right corner).
   - Tap the required node.



If the node is on the finished grade model, the name of the node would be ER, for example. If the node being selected is on the subgrade adjusted surface, it would be named ER-0.250; meaning it is the ER node, but it has a -0.250 m offset. A subgrade can be entered by accessing Road Settings through **Settings**/ **Roads** or by tapping the Road Settings icon in the upper right side and entering a subgrade offset value.



If you are navigating up station, the normal cross section view is displayed left to right as expected. If however, you are navigating down station, then the cross section would

normally be back to front. To reverse the view of the section, select the **Home menu** / **Settings**/ **Roads**, and change the view setting to up station or down station.

5. Let the software guide you to the point to be staked on the selected feature using the values in the info bar. To help you find the point, the guide arrow needs to point up the screen to show that you are traveling directly toward the point. You can turn the guide arrow off and on in the **Design** tab of the **Map Options** screen. The information bars at the top of the screen can be customized using the **Info Bar/Panel** option in the **Home menu** / **Settings**. By default, the info bar shows the design elevation for the point, the amount of cut or fill required to get to that elevation, and how far and in what direction you need to travel to get to the point. You can scroll through the different values using the gray arrows on either side of the info bar. The default map view has the direction north pointing up. To change this so that the direction you are walking is pointing at you, change the map rotation in **Map Options** screen. You can turn on a cut/fill lightbar in the top or bottom panels on the left to graphically show the cut and fill.
6. When you are close to the stakeout point, the software switches into the Fine Stake mode. Additional guidance arrows appear on the top right corner of the map to indicate the remaining distance in each direction. The screen is oriented to the last moving direction before the Fine Stake mode was activated if map rotation in travel direction was selected.
   - When staking with a GNSS, the fine navigation arrows are displayed in a north "up" orientation or based on the direction of approach when the fine stake arrows first appear. Change this behavior with the **Stakeout Guide** option in the **Design** tab of the **Map Options** screen.
   - For staking performed with a total station, the fine navigation arrows are oriented depending on the connection method to the total station.
   - For Bluetooth and cable connections, the directions are as if you were standing behind the total station looking towards the point.
   - For radio connections, the directions are as if you were standing at the prism pole looking towards the total station.
7. After tapping the **Measure** icon, a stakeout report appears. The software creates a Stake Marker report, including the Cross-Section Storyboard to write on the stake in the **Report** tab. A graphical diagram shows how to put an elevation mark on the stake. The software does all the calculations for you. The way that the software calculates the elevation mark and cut/fill depends on the stakeout settings in **Settings**/ **Stakeout**. The software remembers which tab of the stakeout report was last viewed and opens the same tab after staking the next point.

Instead of staking a certain station, you can also stake an alignment at random stations using these buttons on the bottom right of the status bar:

Tap...	to...
	stake at fixed intervals starting at a certain station.
	stake at random intervals somewhere along the alignment, based on the perpendicular location from the centerline.

8. Set the **Auto advance option** field to one of the following options using the **Road Settings** dialog in the **Home menu** / **Settings** / **Roads** or by tapping  :

To...	select...
automatically move to the next station	**To next station**.
automatically move to the previous station	**To previous station**.
manually move to the next or previous station	**No**. This option maintains the current station between points, and enables you to increase or decrease the station when you are ready.
not move a station	Depending on the settings for Station Interval, the Siteworks software will or will not advance to the next station.

9. To set whether the next auto-advanced station will be on a tangent point in the Horizontal and/or Vertical Alignment of the roadway, in the **Tangent Point** option, select

To...	select...
stop at Horizontal tangent points	Horizontal only.
stop at Vertical tangent points	Vertical only.
stop at Horizontal and Vertical tangent points	Horizontal and Vertical.
not stop at any tangent points	None.

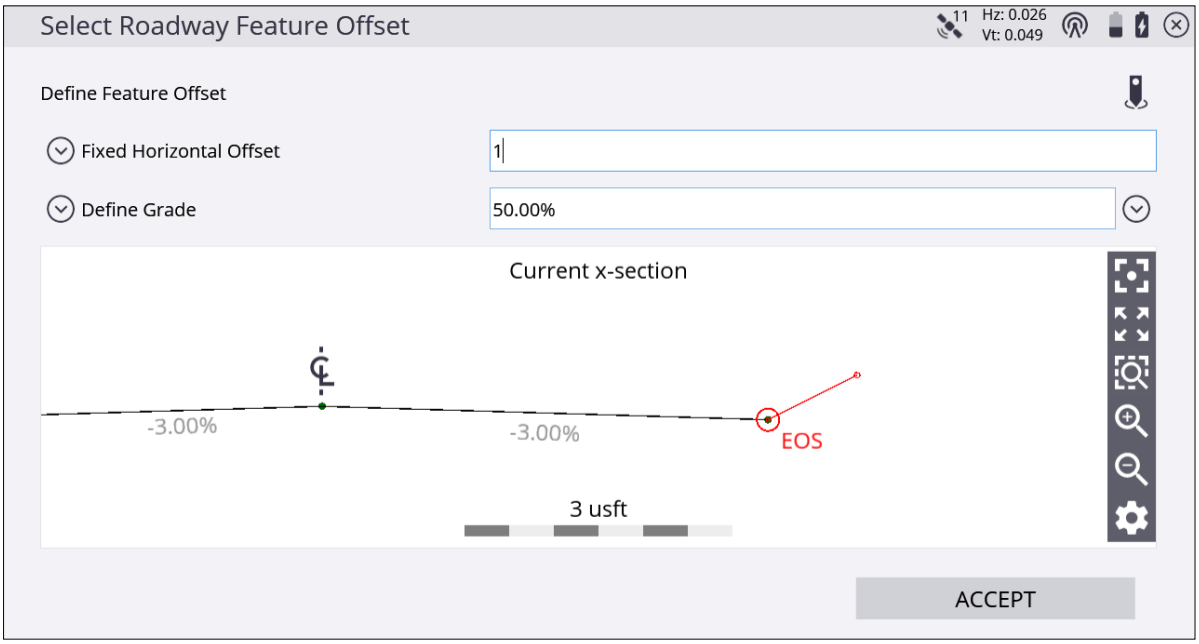
Simple feature offset

When staking a feature, an offset is usually applied. The software is extremely flexible in the way that it enables you to specify an offset.

After selecting roadway feature node (see Step 4 in Roadway feature staking, page 115) tap the button with the stake and the two arrows (roadway feature offset icon) on the top right of the screen:

 .

The **Select Roadway Feature Offset** screen appears. The second line enables you to specify the fixed horizontal offset or select a random horizontal offset. When specifying a fixed offset, a red line and a circle shows you where that offset point is. When specifying a random offset, you can stake anywhere along the cross section; the software displays the results to the point you are at:

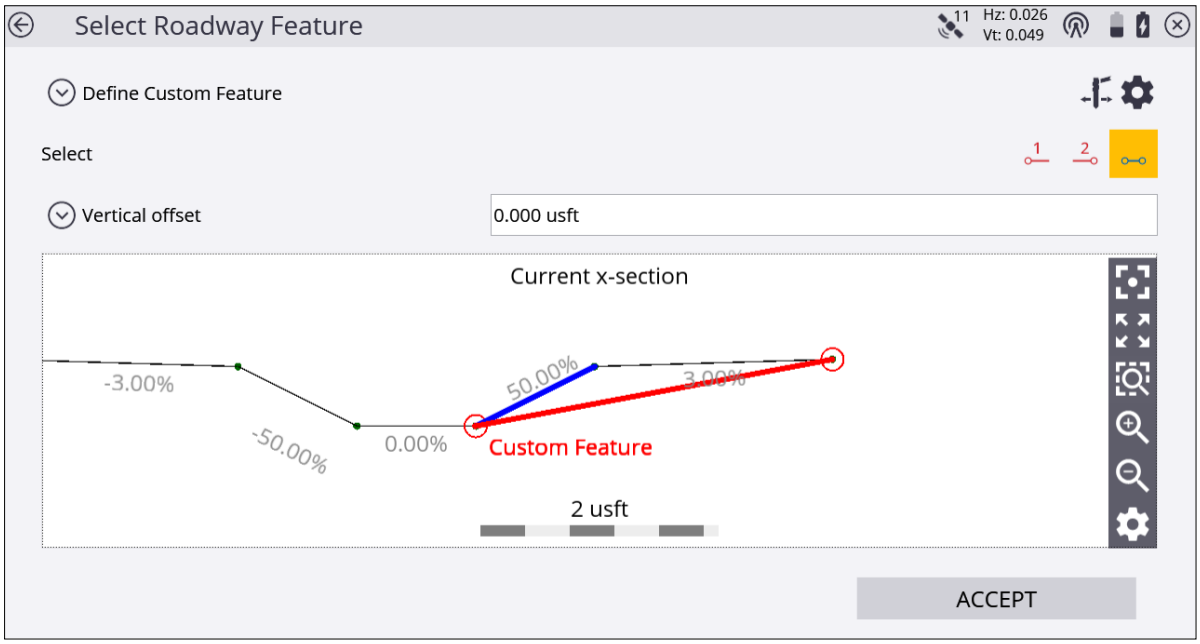


- The third line enables you to specify the kind of offset you want to use from the choices of Define Grade, Select Adjacent Segments, Select Dual Segments. Define Grade enables you to specify a slope for the offset. A grade of 0.000% is horizontal.
- The stake icon  on the top right side of the screen enables you to apply the offset from a different feature node instead of the selected feature node itself.

Custom feature staking

Instead of staking a roadway feature node, you can define your own custom feature on a cross section. One example could be where the subgrade intersects with the tie slope. Follow these steps:

1. While in the **Select Roadway Feature** screen, select the **Define Custom Feature** option from the drop-down menu:

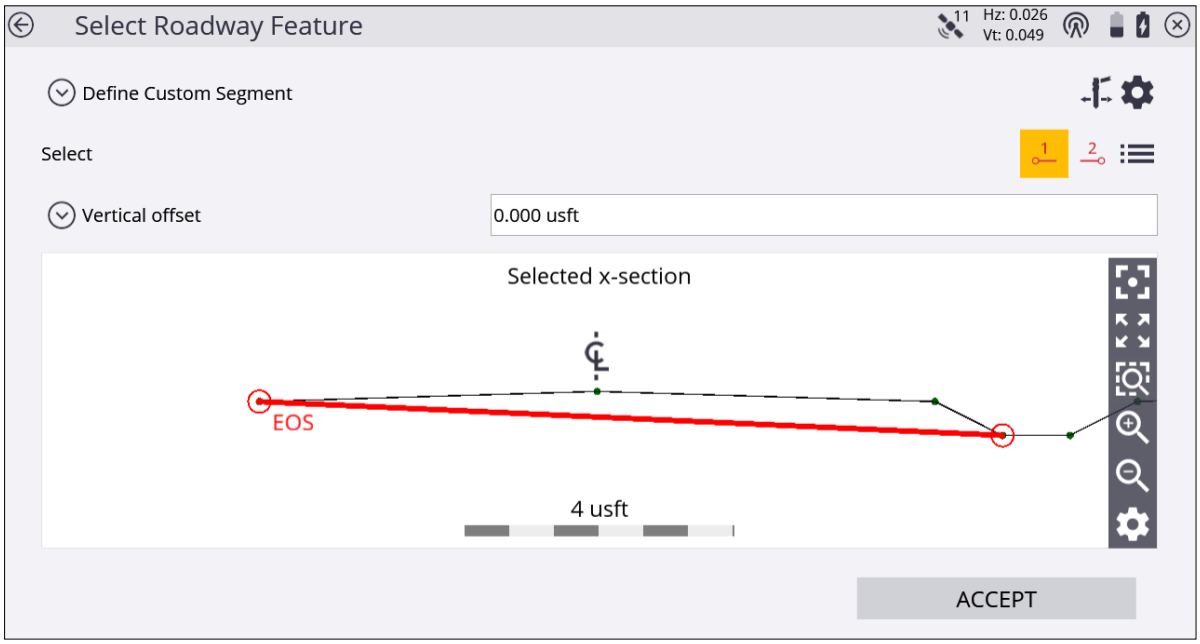


2. Select two feature nodes that will form the subgrade. You can access subgrade adjustments of the finished grade from the map view during stakeout or by tapping on the Road Settings gear icon in the upper right side of the screen (access the **Road** settings through **Settings**, and then enter the amount and direction for the offset).
3. Select a segment (e.g., the tie slope). If required, enter a vertical offset for this segment and if you want to apply the offset vertical or perpendicular.
4. A green circle stake location appears where this subgrade intersects the tie slope. Stake out this point as usual. You can also specify a horizontal offset for the stake the same way as a normal roadway feature.

Custom segment staking

Instead of staking a roadway feature node, you can define a custom segment on a cross section. A segment is defined as the surface between two roadway feature nodes with an optional offset to reflect a subgrade. One example could be where a dam of a road gets built in multiple layers without having the complete shape of the finished grade. With this layered lift function, you can do the following:

1. While in the **Select Roadway Feature** screen, select the **Define Custom Segment** option from the drop-down menu.
2. Select two feature nodes that will form the subgrade. If required, enter a vertical offset for this segment and if you want to apply a vertical or perpendicular offset.



3. You will get guidance to the first point you have selected. Stake out this point as usual. You can also specify a horizontal offset for the stake the same way as for a normal roadway feature. You can also enter a random offset and apply the grade of the roadway and then stake this layer infinitely to the left or to the right and get the cut/fill to this segment.

Catch point staking

Catch point staking from the **Road** menu operates in an almost identical way to the catch point staking associated with the side slope function. The only difference is that the tie slopes are defined in the road model itself, and are automatically displayed. In a typical road model, there are at least two tie slopes; one for the right and one for the left side of the road. In the case of a divided highway, it is common to find four tie slopes, one for the left and right of each of the two traveled ways.

1. From the **Measurement** screen, tap the **Home** menu and then tap **Stake**.
2. Select a road using the list at the top right of the screen and then select an alignment in the **Road** tab. Alternatively, select the road alignment directly from the map or by tapping and holding and selecting a roadway from the pop-up menu. If there is more than one object available in this area, a list of different objects will appear where you can specify a selection.
3. Select a stake method catch point and click accept.
4. Enter a station or select one from the map at which to stake out the road feature. A cross section of the roadway appears unless no tie slopes were defined for this cross section. You can then choose to extend the outer segments of the road and use them as tie slope instead. Other options include defining a tie slope or choosing another station with tie slopes.
5. Select if you want to stake the tie slope of left or the right side of the road by tapping on the tie slope itself or using down arrow on the top right. The selected tie slopes as defined in the road model (a cut slope shows in red and a fill slope shows in blue) will be highlighted.

In the top line of the display, the roadway that the tie slope is associated with is displayed. In the example above, there is only one roadway in the selected road job, and that was not named—hence Roadway 0 is displayed.

You can redefine the current tie slope while you are staking out. For example, the design may specify a 1:3 cut, but when you stake it out you find that the point is outside the site limits, so you change the tie slope to a 1:2.5 cut. You can also project the tie slope from any node on the cross section.

From the list, select one of the following options:

- Original cut/full
- Key-in cut/fill (%) – Enter the new value and then select the node to apply it from
- Key-in cut/fill (rise:run) – Enter the new value and then select the node to apply it from
- Key-in cut/fill (run:rise) – Enter the new value and then select the node to apply it from
- No cut/fill tie – If you select this option, the tie slopes disappear

It is also possible to offset the hinge point by clicking the stake icon on the top right of the screen.

6. Navigate to the correct station using the Ahead/Backward values in the info bar. Then toggle the cross section view using the bottom icon in the map control bar and walk toward the tie-slope you are going to stake.
7. The software calculates the intersection between this recreated surface and the tie-

slope and provides you with guidance to this point by extending this surface toward the slope. This point is dynamic and changes depending on the way you walk and how the terrain is formed. Use inward/outward to find the actual catch point, but also keep an eye on the station.

Once you are in proximity of the point where the tie-slope intersects with the existing ground, tap the **Measure** icon.

The software creates a **Stake Marker** report. A graphical diagram shows how to put an elevation mark on the stake. Depending on the settings for catch point marking in the **Home** menu / **Settings**/ **Catch Point marking**, you will get guidance to one or two more stakes which mark the point. Single Stake, Dual Stake, and Batter Rail methods are available. The placement and marking of these stakes is completely guided. The software remembers which tab of the stakeout report was last viewed and opens the same tab after staking the next point.

8. Instead of staking a certain station you can also stake a line at random stations using these buttons on the bottom right in the status bar:

Tap...	to...
	stake at fixed intervals starting at a certain station
	stake at random intervals somewhere along the line

Location on Surface

Location on Surface is a very basic method of staking a road. You can walk over the road surface and have the display showing station, offset, and cut/fill to this road surface or navigate to a certain station and offset of this road.

1. From the **Map** screen, tap **Home** / **Stake**.
2. Select a road using the list at the top right of the screen and then select an alignment in the **Road** tab. Alternatively, select the road alignment directly from the map. If there is more than one object available in this area, a list of different objects will appear where you can specify the selection.
3. Select **Location** as the stake method from the drop-down menu and tap **ACCEPT**.
4. Enter a station and offset into the boxes or select one by tapping a location on the screen. You can also overwrite the design elevation at this station by selecting **Custom Elevation** by tapping the down arrow in the upper left side of the screen.
5. The map view then guides you to the point. To help you find the point, the guide arrow needs to point up the screen to show that you are traveling directly toward the point. You can turn the guide arrow off and on in the **Design** tab of the **Map Options** screen. The information bars at the top of the screen can be customized using the **Configure Info Bars** option in **Settings**/ **Info Bar**/ **Panel**. The display below shows the design elevation for the point, the amount of cut or fill required to get to that elevation, and how far and in what direction you need to travel to get to the point. The default map view has the direction north pointing up. You can change this so that the direction you are walking is pointing up by changing the map rotation in map options. A cut/fill light bar can be turned on in the top or bottom panels on the left to graphically show the cut and fill.
6. When you are close to the selected line, the software switches into the Fine Stake mode. Additional guidance arrows appear on the top right corner of the map to indicate the remaining distance in each direction. The screen is oriented to the last moving direction before the Fine Stake mode was activated if map rotation in travel direction was selected.
   - When staking with a GNSS, the fine navigation arrows are displayed in a north "up" orientation.
   - For staking performed with a total station, the fine navigation arrows are oriented depending on the connection method to the total station.
   - For Bluetooth and cable connections, the directions are as if you were standing behind the total station looking towards the point.

- For radio connections, the directions are as if you were standing at the prism pole looking towards the total station.

7. After tapping the **Measure** icon, a stakeout report appears. The software creates a Stake Marker report. A graphical diagram indicates how to put an elevation mark on the stake. The software does all the calculations for you. The way the software calculates the elevation mark and cut/fill depends on the stakeout settings in the **Home** menu. The software remembers which tab of the stakeout report was last viewed and opens the same tab after staking the next point.
8. Instead of staking a certain station, you can stake a line at random stations using these buttons on the bottom right of the status bar:

Tap...	to...
	stake at fixed intervals starting at a certain station.
	stake at random intervals somewhere along the line.

To toggle between Feature, Location, and Catch point staking, tap the icon in the bottom of the status bar on the right hand side of the main panel:

Icon	Description
	Feature
	Location on Road Surface
	Catch point

9

Measuring with GPS

To perform these measurement tasks, you need the Siteworks software running on a controller that is either connected to a positioning device like a GNSS system or a Total Station. This chapter explains how to set up and use the different components of an RTK GNSS system.

Setting up the GPS base station

For Precision RTK (Real-Time Kinematic) GNSS operations, two main components are generally required, a base station and a rover receiver. The two components are connected by radio (450 MHz or 900 MHz frequency bands), Wi-Fi, or through several different types of Internet-based communication protocols over which RTK corrections are transmitted from the base station to the rover receiver. The base station is set up in a fixed location where it tracks the satellites of the GPS and, optionally, the GLONASS, Beidu, and QZSS constellations. The rover receiver moves around the jobsite on a pole, backpack, vehicle, or earthmoving machine.

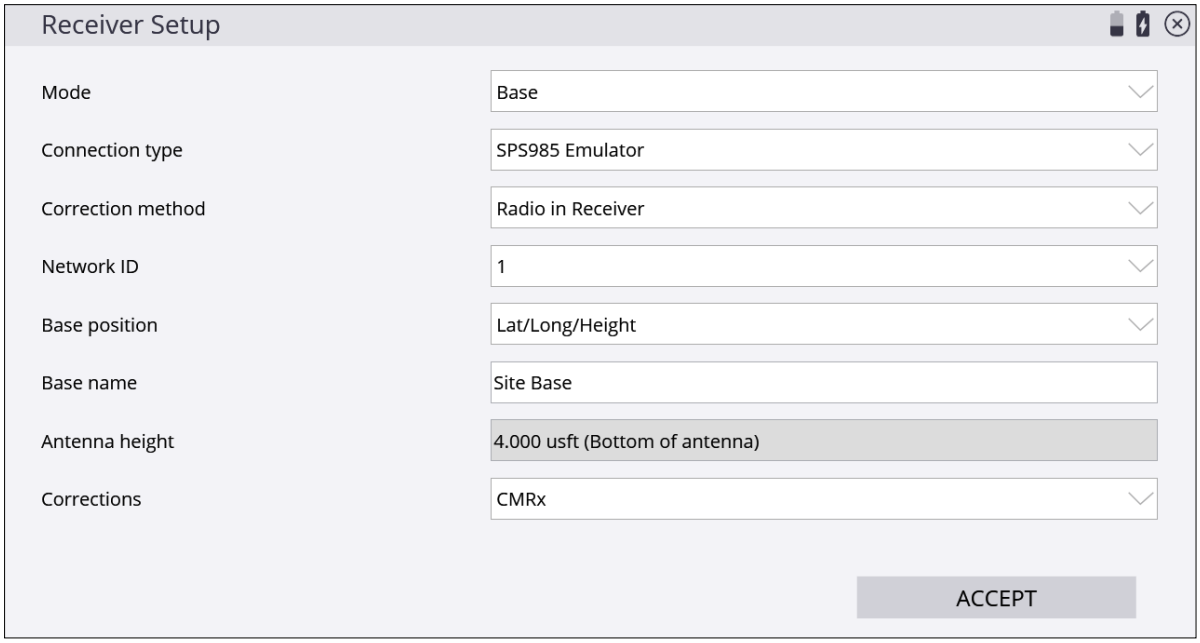
The SPS Modular GPS (such as the SPS855) receivers have a front panel and keypad that enables the base station receiver to be initialized and set up without needing a controller; however, this guide focuses on using the Siteworks software to correctly set up the base station.

The base station setup process remembers how the previous setup was made. It automatically reconnects the components, selects the appropriate radio channel or network number used previously, and then automatically starts to transmit GPS positions. If the base station has never been previously established on the site, or if on the last base station setup a cellphone was used to broadcast corrections, you must completely set up the base station again using the options in the Connection Typeand Connection Methoddialog. In these situations, the Setup radio only option is not available.

When using the SPS Smart GNSS antenna or SPS Modular GNSS receivers, which use AutoBase™ technology, once a base station has been established for the first time, if nothing was changed between base setups, you can simply set up the receiver at the exact same base station location and then switch it on. Using AutoBase technology, the receiver reloads all appropriate data, makes all appropriate connections, and then starts to transmit corrections on the last used radio channel or network number. This eliminates the need to use a controller with the Siteworks software to set up the base station each day. If you want the SPS GNSS receivers (SPS Smart GNSS antennas, and the SPS Modular GNSS receivers) to operate in AutoBase mode, you must name each base station with a different name; otherwise the AutoBase mode will not work. The AutoBase technology can be configured in the web interface. Note that you must set up the base station at the exact same antenna height each time for the AutoBase technology to correctly work (that is, the base station is set up on a dedicated rigid pole). If different base antenna heights are used, for example on a tripod that is broken down and setup each day, then the base must be set up each time in Siteworks software using the following process to ensure the correct heights and settings are configured.

To set up a GNSS base station:

1. From the **Home** menu/ **Project Setup**, select **connect device** and then tap **GPS**.



2. Set the **Mode** field to Base and then answer each question to complete the base station setup. In the **Base position** field there are several options on where the base is setup.
   - Control Point – An existing control point present on the site and imported into the Site folder.
   - Unknown Position – For a site with no known coordinates; uses an autonomous "HERE" position as the base location.
   - Local Coordinate – A site that has control points in local site coordinates.
   - Lat/Long/Height – Enables you to enter a known latitude/longitude and height, or use an autonomous "HERE" position.
   - Base Anywhere – Allows the base to be set up anywhere on site, not on a control point, and will use an autonomous HERE position as its location. Note this type of base setup should only be used with a rover receiver also configured in BaseAnywhere mode. For more information, see page 130.

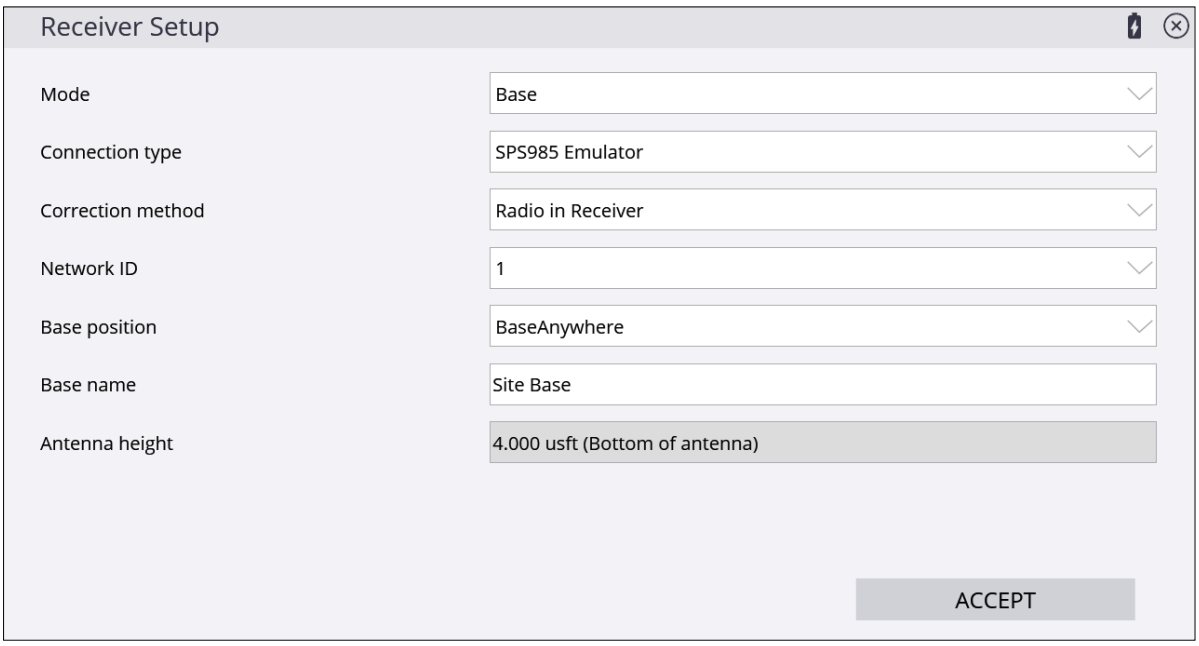
You only need to set up the base station once; the software remembers your settings and prompts you to use the same ones the next time you set up the base station within a given site. If necessary, you can then change any of the base settings such as the control point the base station is set up on or the antenna height. For new sites, the base will need to be set up from the beginning.

BaseAnywhere base station setup

BaseAnywhere is a feature that enables a base station to be set up anywhere on the job site, including on the roof of a vehicle, a range pole with a bipod, or any other stable platform. It does not require the base station to be set up on an established control point and greatly simplifies and quickens the ability to get started with an RTK GNSS survey on a site. This feature is generally intended for use by smaller contractors on sites that do not have existing survey control points.

It works by setting the base station into AutoBase HERE Always mode, which means that each time the base station is turned on or power-cycled, the base will determine its autonomous position and begin to broadcast corrections on the last used radio or Wi-Fi settings. This base station setup configuration must be used with the associated BaseAnywhere and Bench My Rover configuration to ensure that correct positions are being used, as the "Here" position of the base station will change each time the base is restarted.

To configure the base receiver to broadcast in BaseAnywhere mode, select BaseAnywhere in the Base position field when setting up the base station:



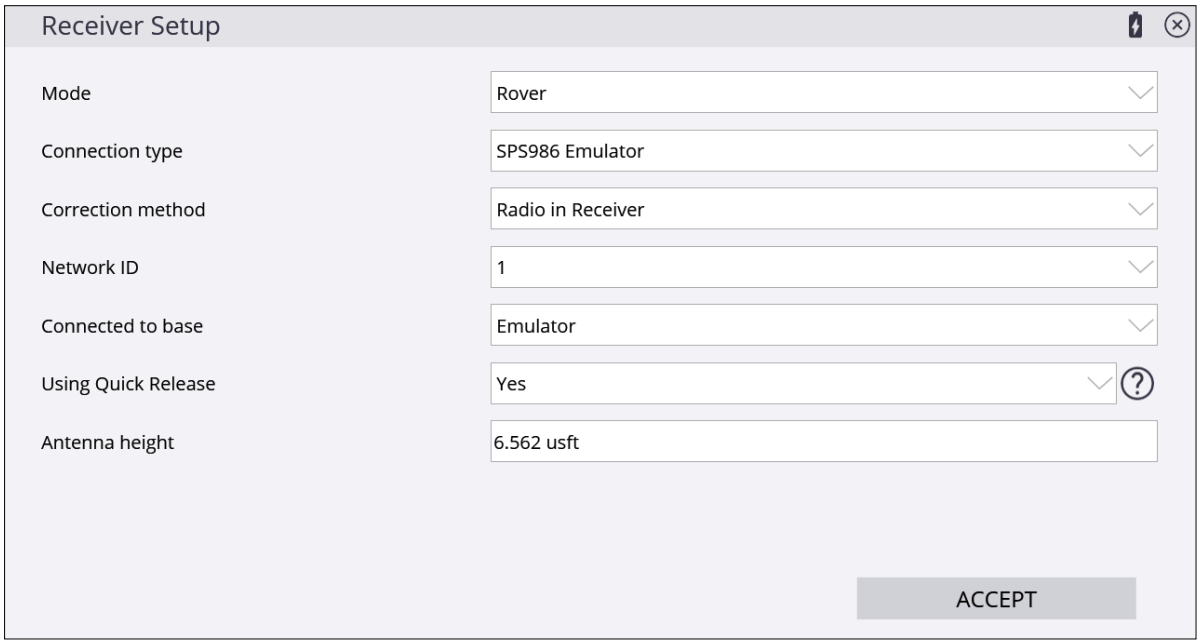
Once this mode for the base station is set there is no need to reconfigure the base station again unless there has been a 15-second restart of the receiver, you want to set up on a known point, or use/change any of the other base setup options. Note that BaseAnywhere base stations can only be set up to broadcast corrections via the receiver's internal radio or Wi-Fi.

Also note that the base name when using BaseAnywhere will default to AUTO000X after a restart of the base receiver, where X is the number of times that the base has been started. So, if the base station has been restarted five times, the base station name will be AUTO0005. The base name broadcast from the base station will not match what was originally entered in the Base Setup screen.

Setting up the GPS rover receiver

To set up the rover receiver:

1. From the **Home** menu, select **Project Setup** / **Connect Device** and then tap **GPS**.



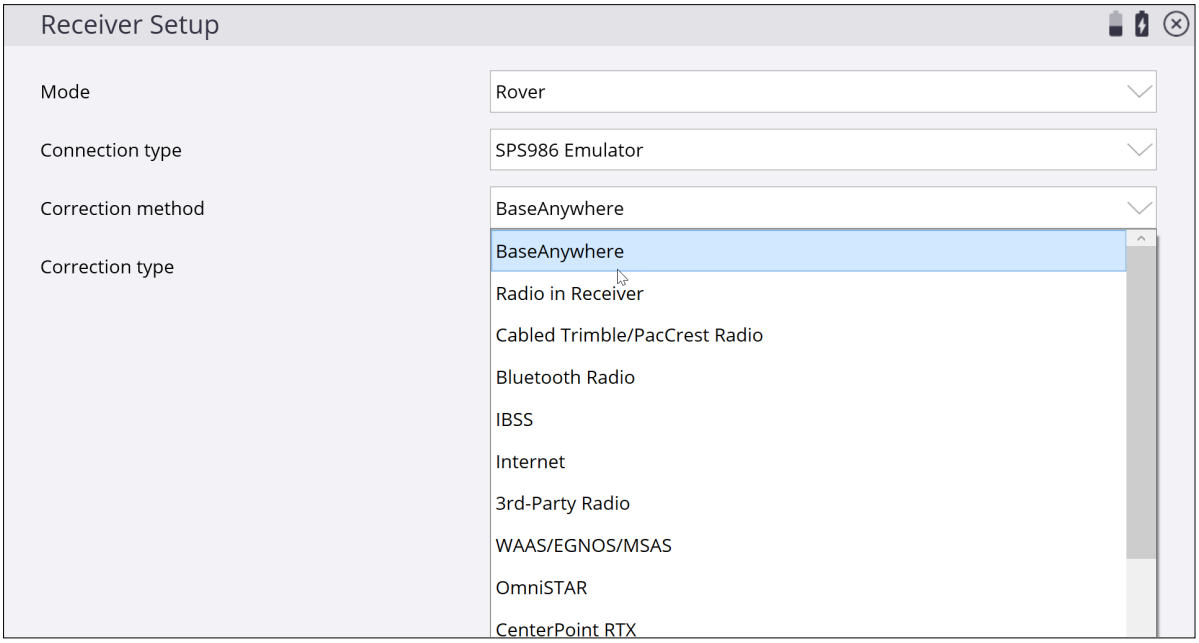
2. Set the **Mode** option to Rover and then answer each question in turn to complete the rover setup.

You should only need to do this once per site; the software remembers your settings and will have them pre-filled for the next time you set up the same rover receiver in the same site. You can change any of the rover settings if necessary. If any of the settings have changed, such as the base station radio channel, the rover will need to be configured from the start for each new site or to connect to a new base.

BaseAnywhere rover receiver setup

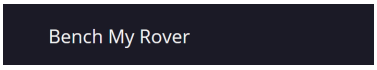
Using BaseAnywhere at the rover receiver involves benching the rover at a known control point so that a proper offset from the base station's autonomous position to a known control point measured at the rover receiver can be determined. After turning on and configuring a BaseAnywhere base station, and configuring the rover for use in BaseAnywhere mode, you are required to bench the rover receiver on a control point before any measurements can be taken. The only exception is for new sites without existing control points that have not yet been calibrated. In this case, you are prompted to calibrate the site using a one-point calibration before any measurements can be taken.

To use BaseAnywhere at the rover, select it from the **Correction method** field:



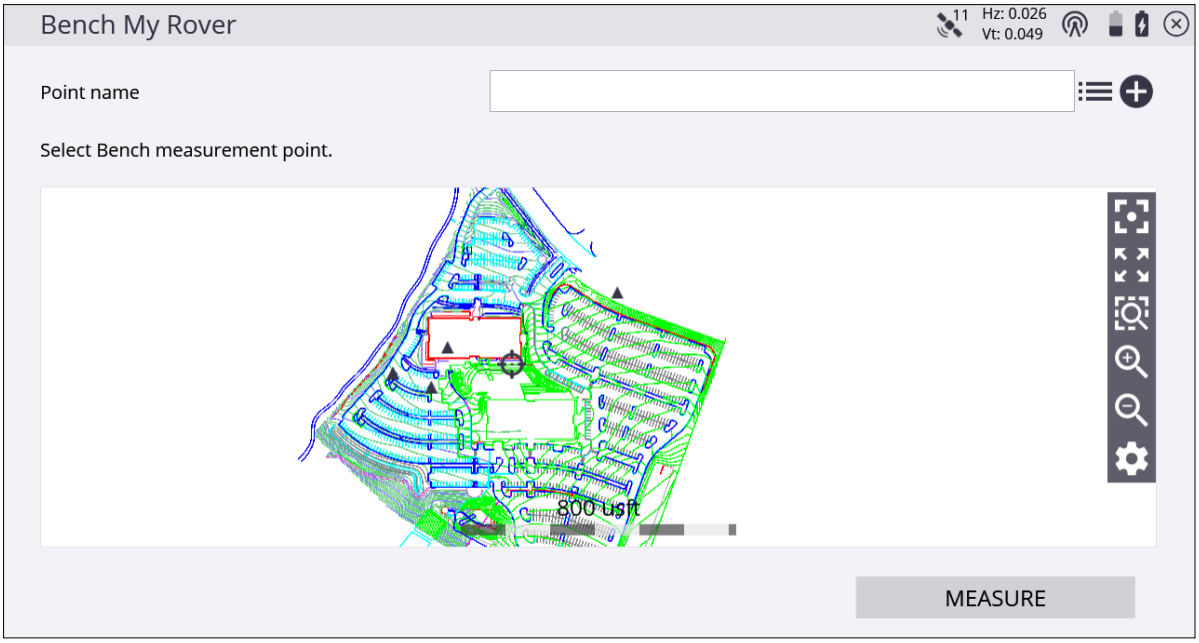
If your site does not contain any control points and has not been calibrated yet, you will be prompted to perform a one-point site calibration, which is the standard Siteworks workflow. After performing the site calibration, you must measure at least one additional control point so that you can successfully perform a system check as part of the Bench My Rover process.

If the site is using an existing coordinate system or you are importing an existing calibration into a new site, you will need to have existing control points in the Site folder so that the Bench My Rover process can be completed.



To access Bench My Rover, tap the **Home** / **Project Setup** / **Bench My Rover**. The icon is only available after a site has been calibrated. When opening an existing calibrated site when using BaseAnywhere, you are taken to the **Bench My Rover** screen before being able to measure any points.

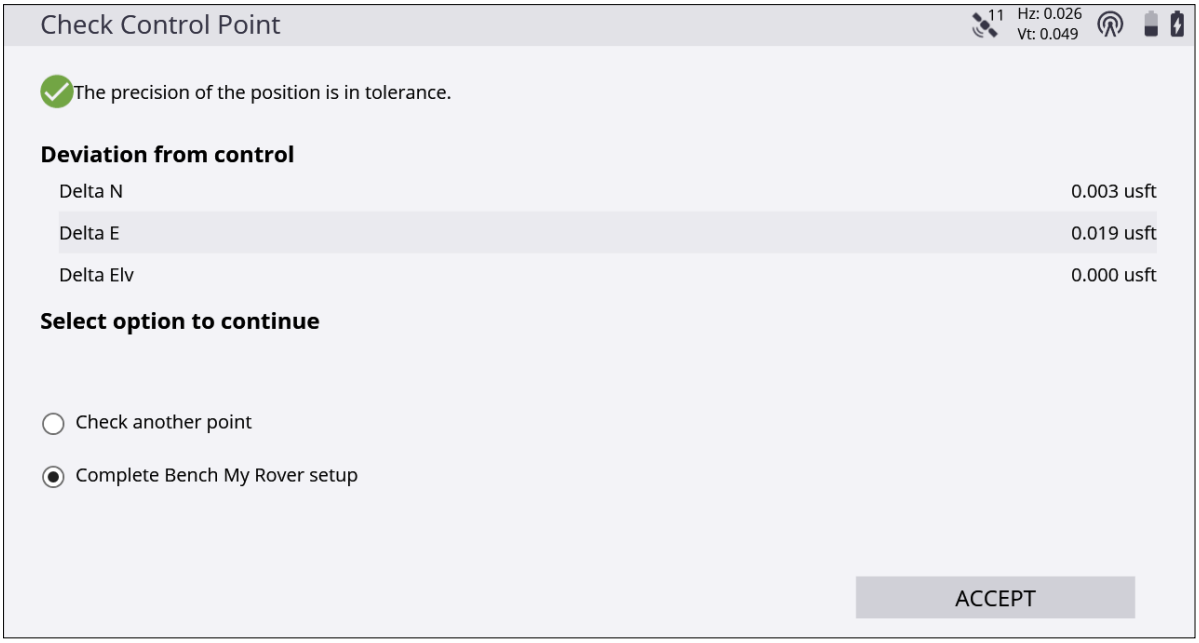
After entering the Bench My Rover workflow, the following screen appears prompting you for a Bench measurement point:

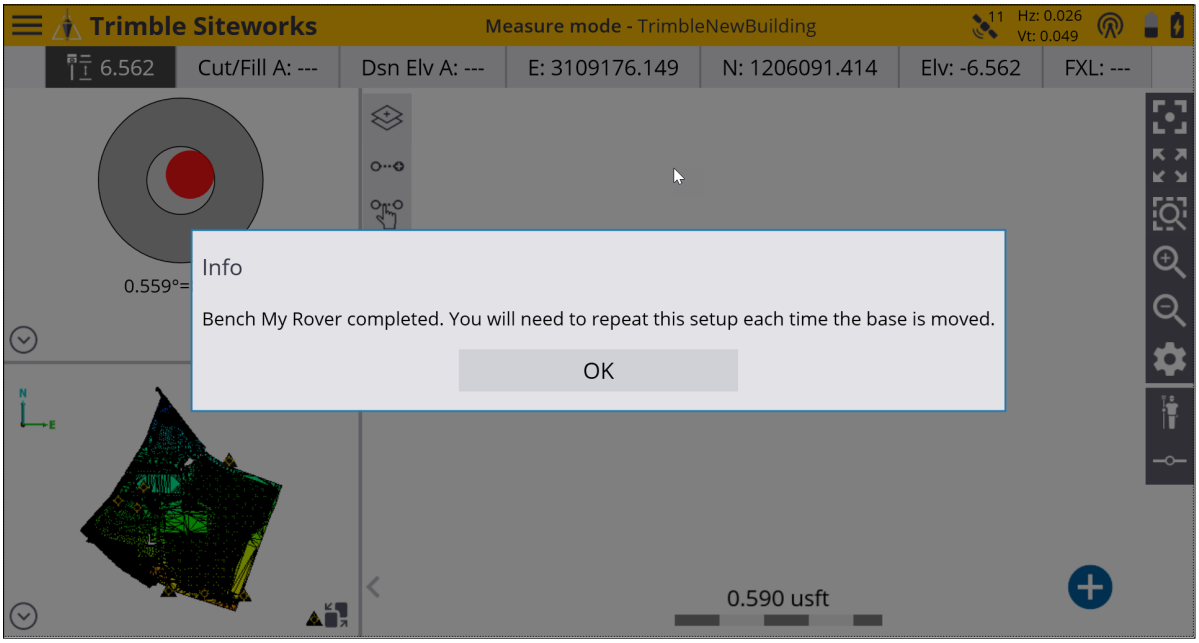


Select the point by doing one of the following:

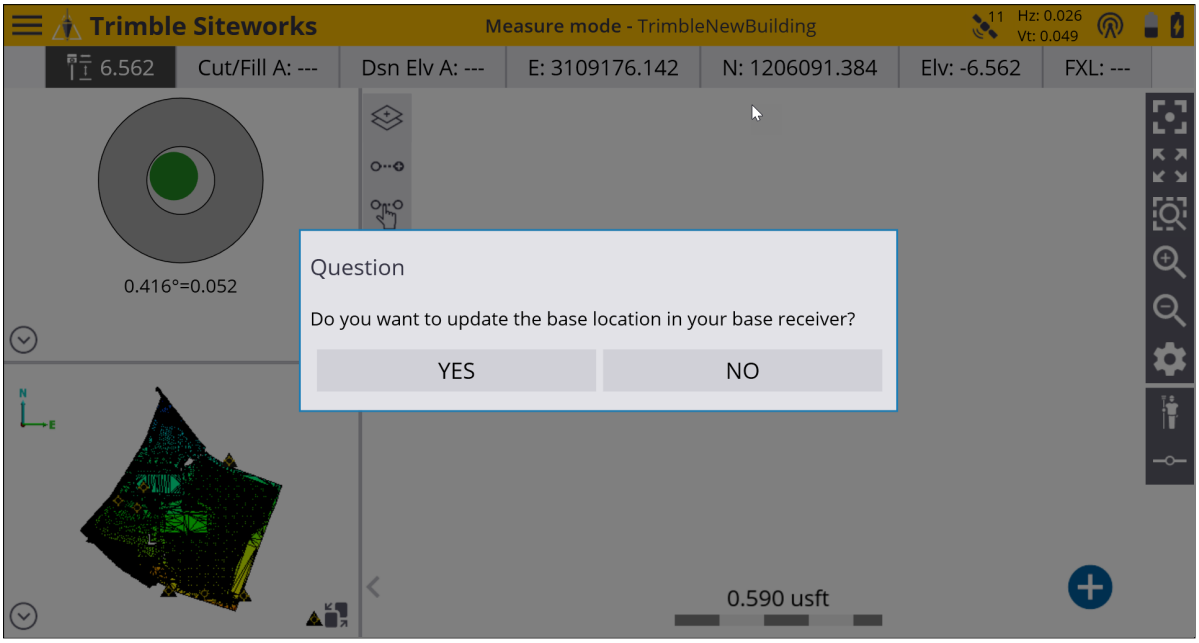
- Tap it on-screen.
- Tap next to the **Point Name** field and select it from the list.
- Tap to enter a new control point.

After performing the Bench My Rover routine, you are required to measure a check point to ensure that the measurements are correct. The software prompts guide you through the check point measurement process. After measuring the check point, you will be shown the tolerances and given the option to select to measure a new check point or complete the Bench My Rover setup. At least one check point is required to be measured in order to use BaseAnywhere to ensure data integrity:

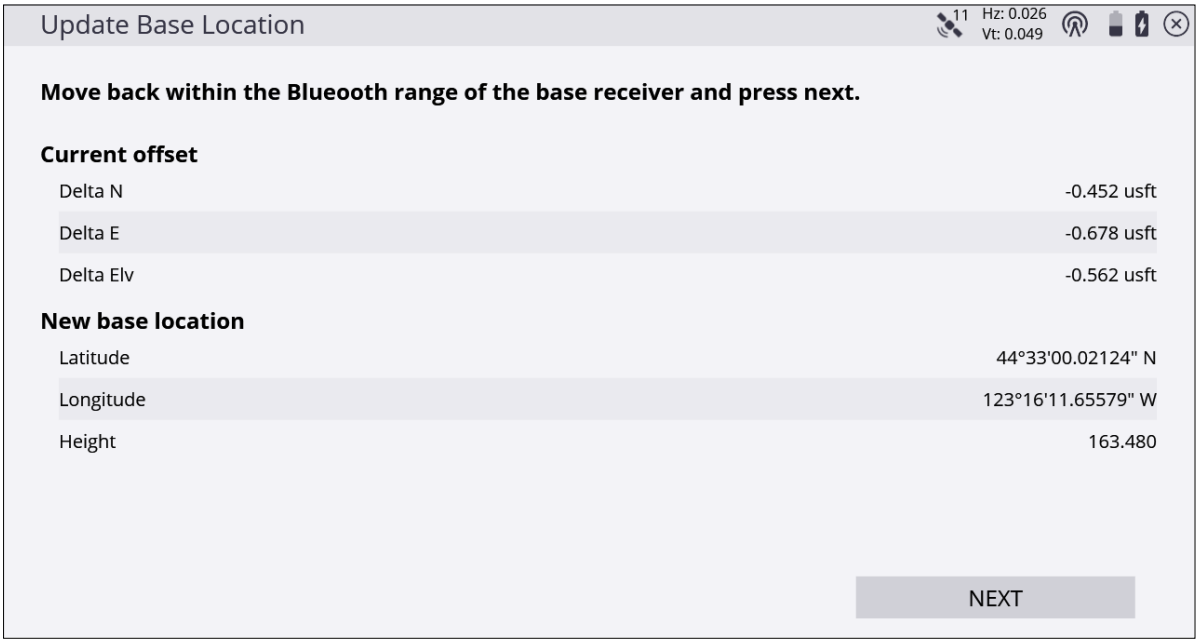

Check Control Point
11 Hz: 0.026 Vt: 0.049
The precision of the position is in tolerance.
Deviation from control
Delta N 0.003 usft
Delta E 0.019 usft
Delta Elv 0.000 usft
Select option to continue
Check another point
Complete Bench My Rover setup
ACCEPT


Trimble Siteworks
Measure mode - TrimbleNewBuilding
11 Hz: 0.026 Vt: 0.049
6.562
Cut/Fill A: ---
Dsn Elv A: ---
E: 3109176.149
N: 1206091.414
Elv: -6.562
FXL: ---
0.559°=
Info
Bench My Rover completed. You will need to repeat this setup each time the base is moved.
OK
0.590 usft

After the Bench My Rover routine is performed, a prompt appears asking if you would like to update the position of the base location:



Selecting Yes will enable an update of the location of the base receiver from its autonomous position to its true RTK-calculated location. This is now the equivalent of having a traditional base setup on a known control point.



Updating the base receiver location this way enables the base, *while operating at its current location only*, to be used for machine control applications and by additional rover receivers on site, without having to perform Bench My Rover routines on each rover. If the

base receiver loses power, is restarted, or moved, a new Bench My Rover routine and updating of the base location must be performed before the base station can again be used by machine control applications or additional rovers.

Trimble recommends that you perform a Bench My Rover routine several times during the day, and any time after the base station may have moved (if for example, the base is placed on the roof of a vehicle). The software will force a Bench My Rover routine if it detects that the base station has been power cycled or if a new base name on the same channel is detected.

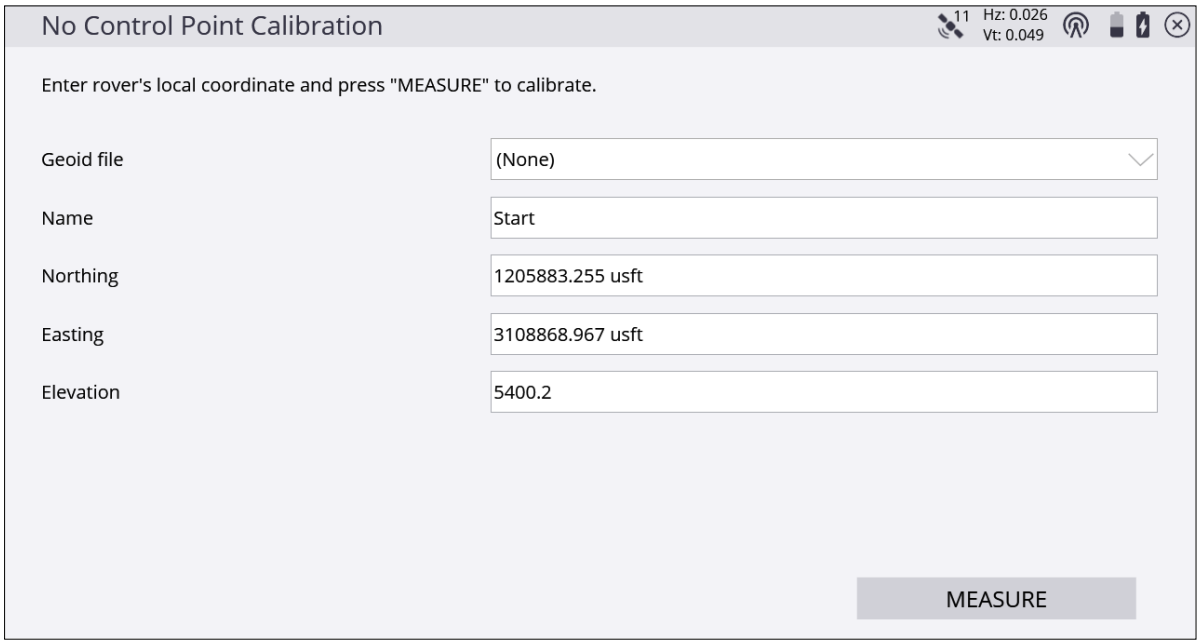
⚠ CAUTION – As with any base station setup, if the base station moves while transmitting corrections you will get incorrect measurements at the rover receiver. If base movement is suspected, performing a Bench My Rover will adjust for any movement in the base station.

Calibrating the site

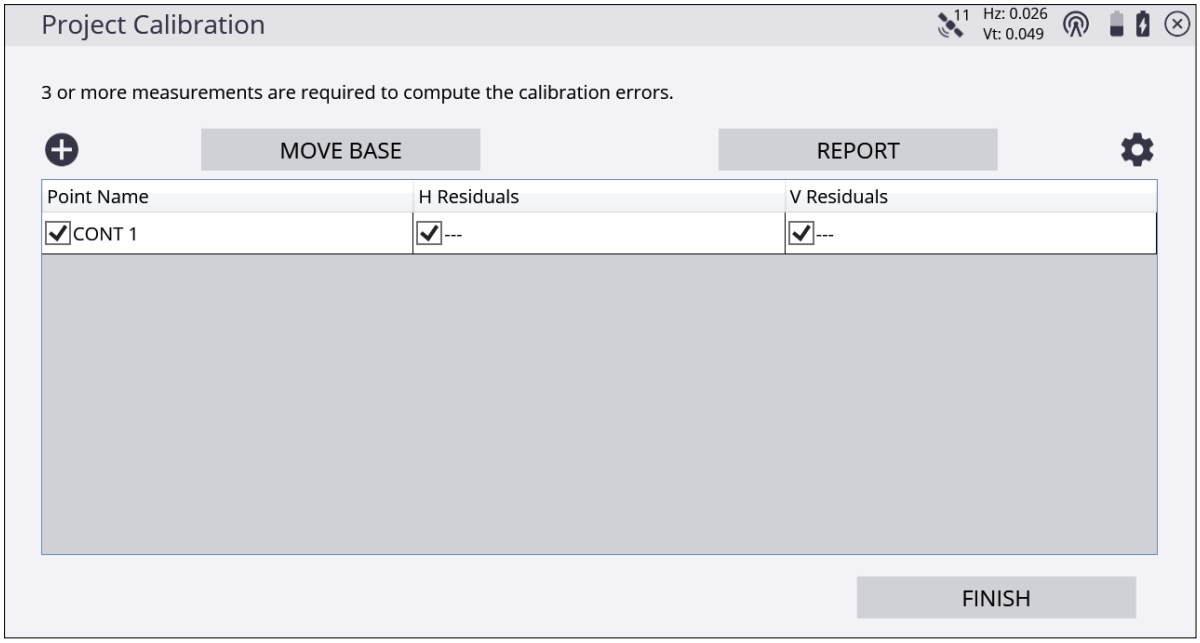
Global Navigation Satellite Systems (GNSS) produce positions in latitude, longitude, and height coordinates. Construction projects are generally designed in northing, easting, and elevation (or X,Y,Z) Cartesian coordinates. A project calibration ties the GNSS positions to the local site coordinate system so that GNSS can be used to measure or stake out on the construction site. The project calibration process involves measuring a number of known control points in the local site coordinate system using a GNSS rover, allowing the software to create pairs of measured latitude, longitude, height, and known control points.

To calibrate the project:

1. From the **Home** menu, select **Project Setup** and then tap **Project Calibration**.
2. If you have no control points in your project, you are prompted to enter a coordinate for where you are standing. The software computes a one-point calibration based on this coordinate and will be oriented without any rotation so that the project's local north direction will be true north:



3. If control points exist in your project, you are prompted to select a control point and then measure the physical point on the ground with your rover receiver. To add a calibration point in the Project Calibration table, tap the plus icon: 



4. When enough control points have been measured, horizontal and vertical residuals will appear on the screen.
5. If these are acceptable, tap **Finish** to end the calibration. The calibration is then used for the project.

The project calibration is stored in a DC file that can be used with other Trimble equipment working on the project. The software can also export the project calibration to a CompactFlash card as a CFG file for use with Trimble GS900 or Trimble Earthworks machine control systems.

The software notifies you after three points have been measured whether the calibration is in or out of tolerance with respect to the calibration tolerances. After each point you have the following options:

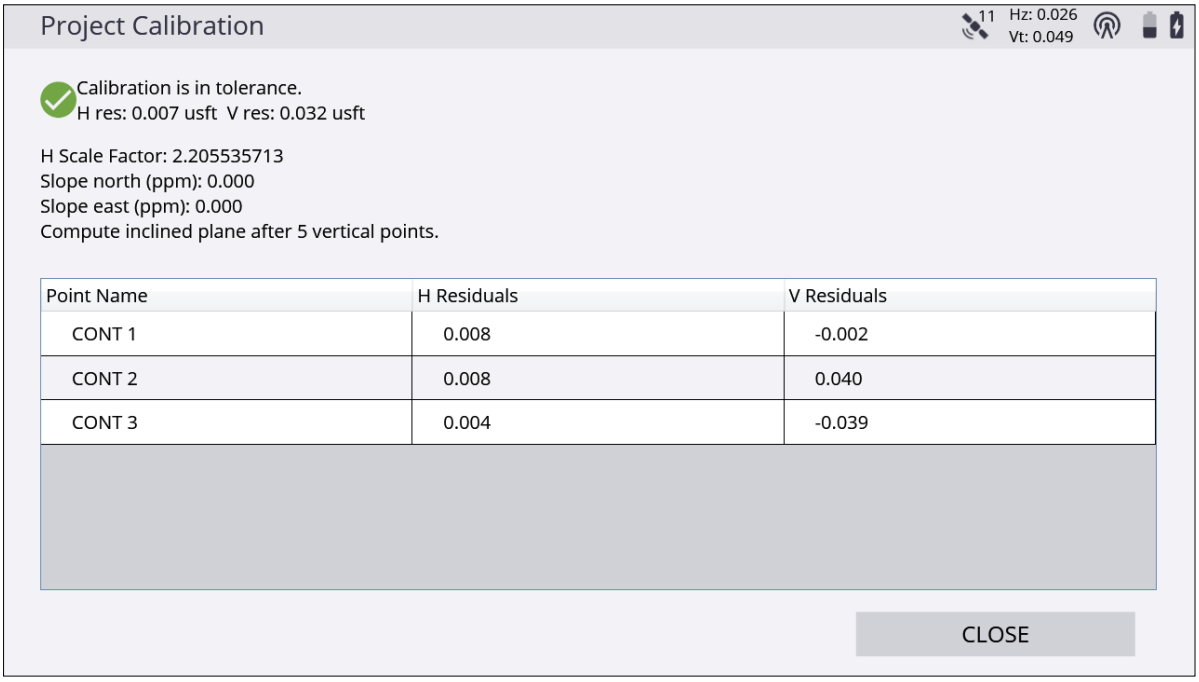
- measure additional points
- retake a point flagged as potentially in error
- save the partial calibration and resume later

The danger of using residuals as the only means of controlling a calibration is that the best precisions can be achieved using the wrong combination of points in the calibration solution. When an inclined plane is in use, this manipulation of residuals can result in a steeper tilt of the plane to best fit the data, resulting in better precisions and an in-tolerance calibration. Monitor the tilt of the plane closely, especially when the geometry of the control points is not strong. Widespread control that covers the entire site is good; narrow-based control around a corridor is not as good. An incorrect tilt of the plane can

result in increasing errors in height as you move away from the center of the controlled area.

Once a project calibration is performed and completed, you cannot change or add to the calibration in that project. To add points to a completed project calibration, start a new project and import the existing DC file from the project which you want to add control points to.

You can review a project calibration completed in Siteworks by tapping the Home / Project Setup / Project Calibration. You are prompted if you would like to review the calibration report. Tap Yes to bring up a table showing the calculated scale factors, point pairs, and residuals:



Tap again in the table to activate the table and enable you to turn off/on individual points, to see the effects on the residuals and scale factors. Note however that this is a display-only feature, therefore any changes made in the Project Calibration Review table are not saved, and the calibration will not be affected.

Performing a two-point calibration

Trimble recommends that wherever possible you use a multi-point calibration of at least three points. Use the two-point calibration method in situations where a baseline of only two control points is available. In the two-point calibration, the first point establishes the position and elevation for the project; the second point establishes the project orientation.

In a two-point calibration, the heights for the project are computed using a simple block shift method that ties all heights to the first measured control point. A two-point calibration is carried out in the same way as a multi-point calibration.

Troubleshooting a site calibration

If a project calibration fails repeatedly, try the following solutions:

- Try a different combination of control points. The software cannot always identify the bad point.
- Start the calibration process again. You may have incorrectly measured a point or points.
- Check the equipment. The source of the error may be as simple as the adjustment of the rod bubble, or a bent rod.

Once the system is set up there are limited sources of error when using RTK GNSS systems. The most common sources of error include:

- A poor project calibration
- Incorrect base antenna height
- Incorrect rover antenna height
- Incorrect selection of the correct antenna type at the base or rover, which causes height errors
- Incorrect location of the base station antenna
- A GPS rod bubble is out of adjustment or the rod is bent

These errors can easily be detected by rechecking the system setup. After starting the rover each day, the software prompts you to recheck the system setup. All recheck system setup operations are logged in the work order report and record files for reference and troubleshooting requirements.

NOTE – The prompt for a system recheck can be turned off by accessing the managers menu by tapping Ctrl+O (letter "oh"). Additionally, the prompt to adjust a published coordinate system with a site calibration can be turned off in this menu.

Measuring a new control point with GNSS

There are several reasons why you may need to measure a new control point during a project and then add the measured position to the Control Point file for the project. Typical cases include:

- When operating the project with a mixture of GNSS and total station equipment; total stations require more control points around the project because of their line-of-sight dependence. Control points can be established very rapidly using GNSS and can be used later by total station crews to establish their position and orientation.
- When operating a site, a base station often needs to be moved closer to the current working location to provide better radio coverage. The base station must be relocated to a known point in the local coordinate system, unless BaseAnywhere is used. Using the Measure control point option ensures that the point is created in the correct location, with the correct coordinates, and guarantees that the project calibration remains valid after the base station is moved.
- When carrying out topographic measurements on a new site before control for the project is established, you may set the base station up in an arbitrary and convenient location and then carry out a single-point calibration. Once completed, measuring three or more control points around the project provides a common set of reference points that can be used later to transform the data measured using the single-point calibration, to the project coordinate system once it is established.

1. From the **Home** menu, tap **Measure** / **Measure Control Point**.
2. Create the control point location using a stake, a hub, or a road nail as required and then mark the stake with the name for the control point, for example, CP3.
3. Set up the GNSS rod over the point and hold it steady using a bipod. The software displays the current GNSS position on the map.
4. When you are ready to take the measurement, tap the **Measure** icon. The measurement process takes approximately 15 seconds to complete. Measuring for 15 seconds takes an average position that increases the accuracy of the computed control point.

When each control point is measured, the software stores the control point data position in the control point (Control.field.csv) file for the site, and also records the measurement data into the record and report files for the work order.

Measuring using xFill technology

The optional xFill™ option for SPS GNSS receivers allows maintaining an accurate positioning solution for an additional four minutes after the main correction data source has been dropped, for example, in a radio blackspot or in an area where no cell coverage is available when measuring with VRS™ technology. During this time the receiver uses L-band satellite-based corrections. If the correction data source is restored during these four minutes, the xFill technology seamlessly drops in and out. Points measured with xFill technology are stamped as such as the accuracy decreases when coming closer to the four-minute mark.

To use the xFill technology, the receiver needs to track the xFill satellite for at least 15 minutes.

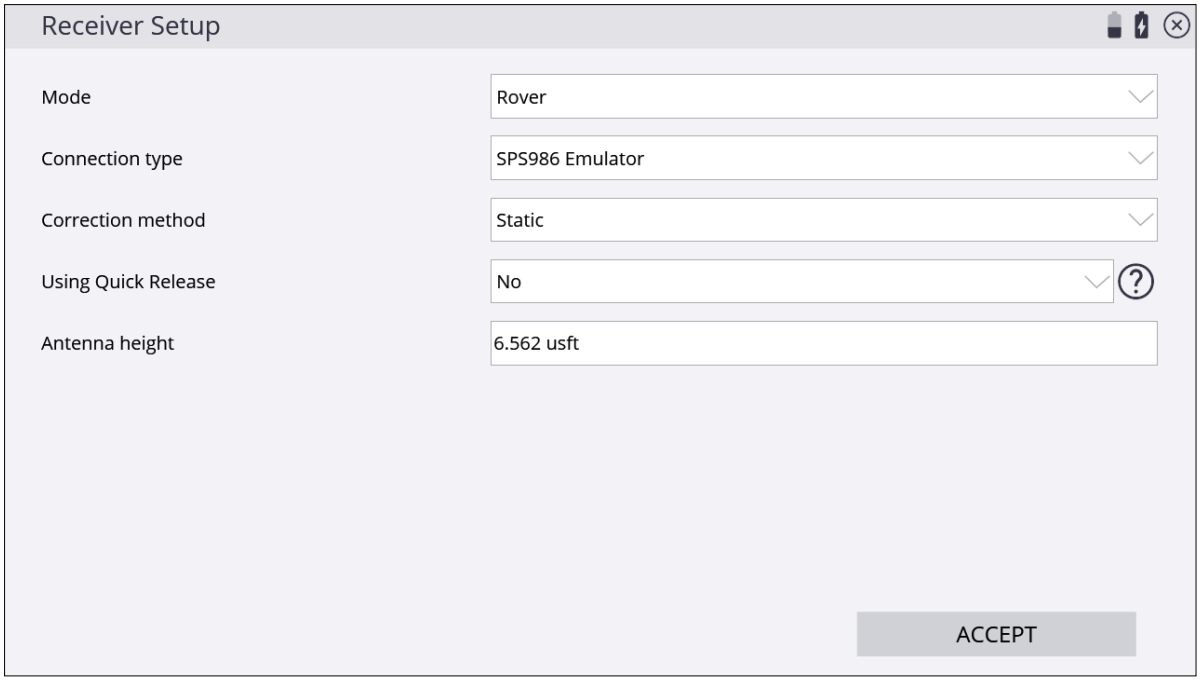
Static measurements

If the receiver has the Data Logging option installed, the software can configure an SPS GNSS rover receiver for static measurements.

Static measurements allow for the averaging of the GNSS results over a user-selected time period, resulting in a more precise position.

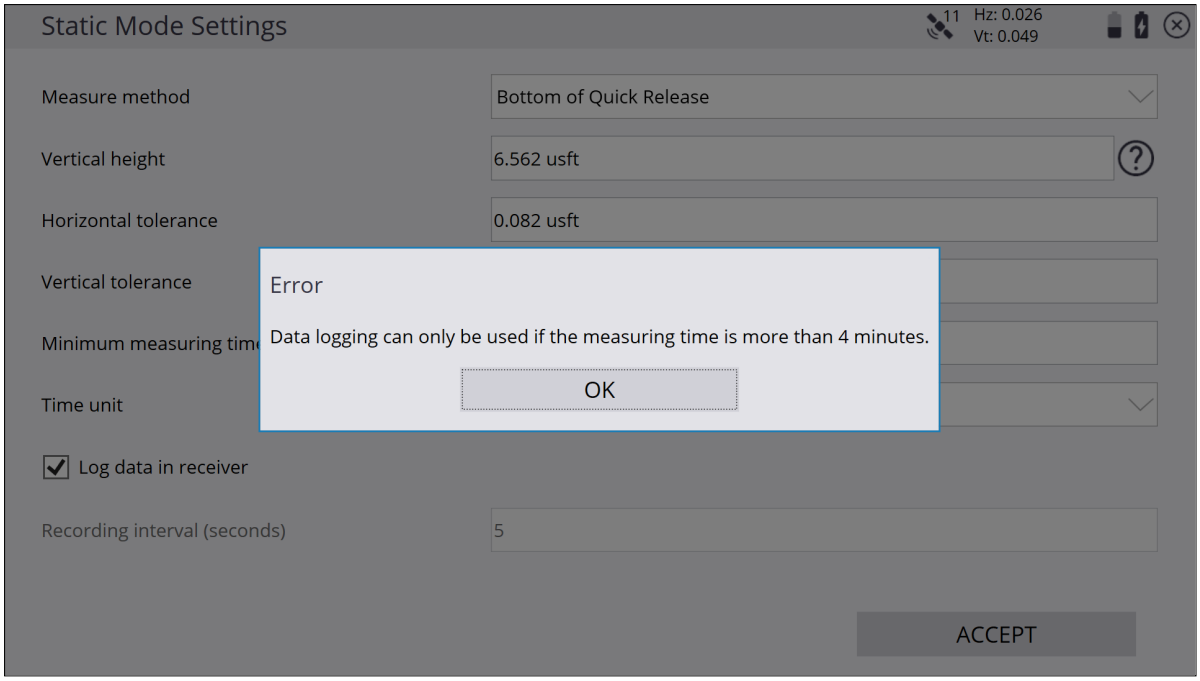
To set up the receiver for static measurements:

1. From the **Home** menu, tap **Project Setup** and then tap **Connect**. The following dialog appears:



2. Set the **Mode** option to Rover and then answer each question that appears to complete the rover setup.
3. After setting up the receiver a message may appear asking if you want to calibrate the site; tap **No**.
4. Change the measurement mode to Static; tap the **Measure mode** icon in the status bar on the right hand side of the main panel and then tap the static icon:

Alternatively, select the **Home** menu / **Settings** / **Measure Mode**. Selecting **Static** opens the following dialog:



5. Complete the options for the antenna height, how long you want to measure for, and select whether or not to store the raw data in the receiver. You need to measure at least four minutes to store RAW data in the receiver, which can then be used for postprocessing.

   If you wish to log the file into the receiver for later download via USB cable or the web interface, enter a point name into this field that will be saved into the .T02 file. This log point name will appear as the point name when the .TO2 file is imported into the Trimble Business Center software. Note that the point name entered in the **Static Mode Settings** screen is different than the point name that is entered after the static measurement is complete. The name entered first into the **Static Mode Settings** screen is only stored in the .T02 file. The name prompted for after the measurement is complete is the point name stored and displayed in the work order in the Siteworks software.

6. When you are ready to start the Static measurement, tap the **Measure** icon and check the **Static Mode Settings**. Then tap **Start**. The software counts down the time that you have entered and then gives you the option to fill in the point name and code and accept and store the measurement.

NOTE – If the receiver loses power, is shut down, or the measurement canceled, no TO2 file containing the raw data will be saved, and all the static data for that point will be lost.

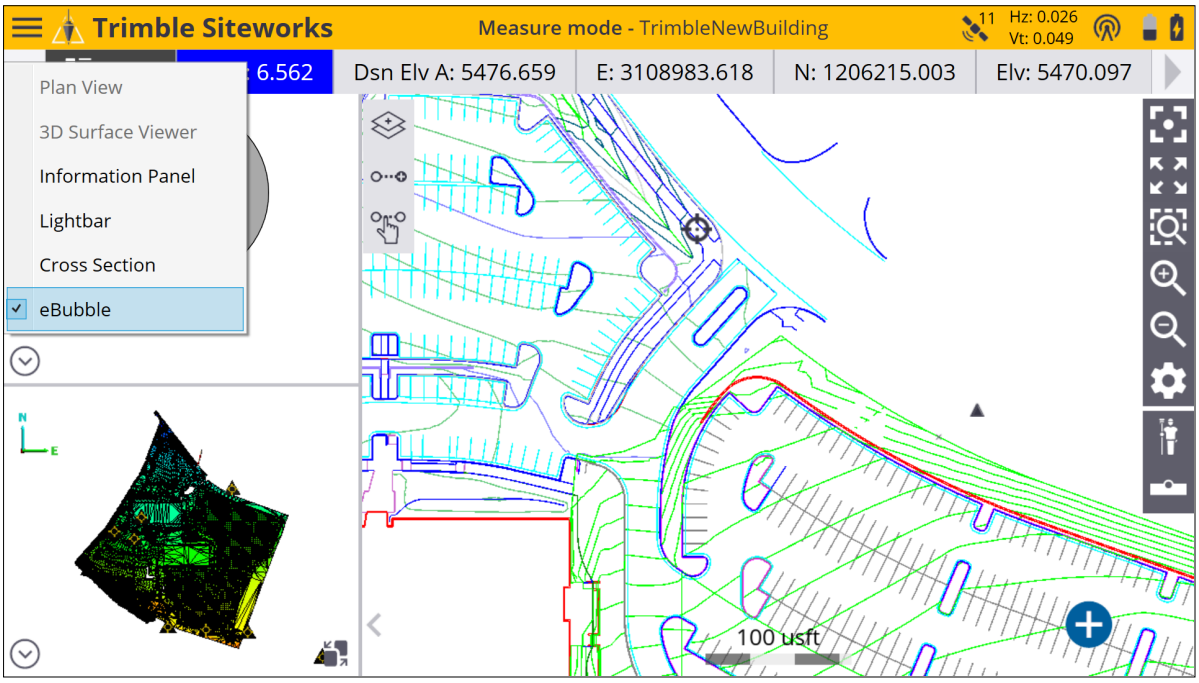
10

eBubble

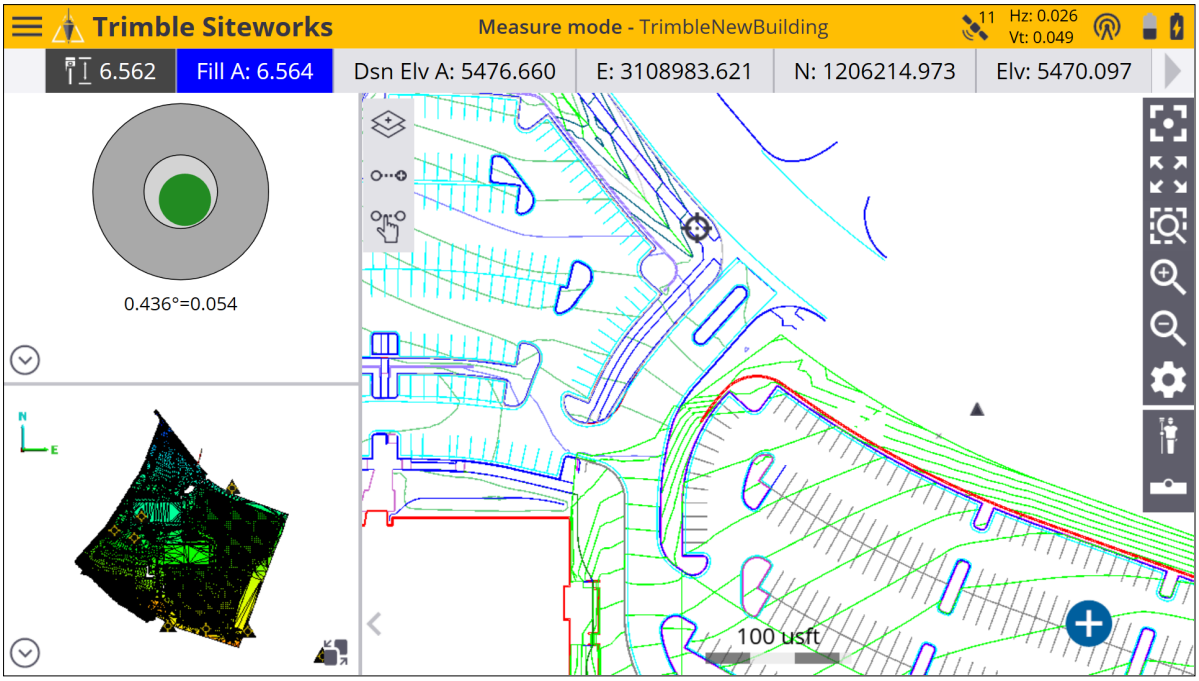
Siteworks software supports eBubble functionality for GNSS receivers that contain the appropriate tilt sensor hardware (currently the Trimble SPS986 and R10 receivers).

eBubble operation

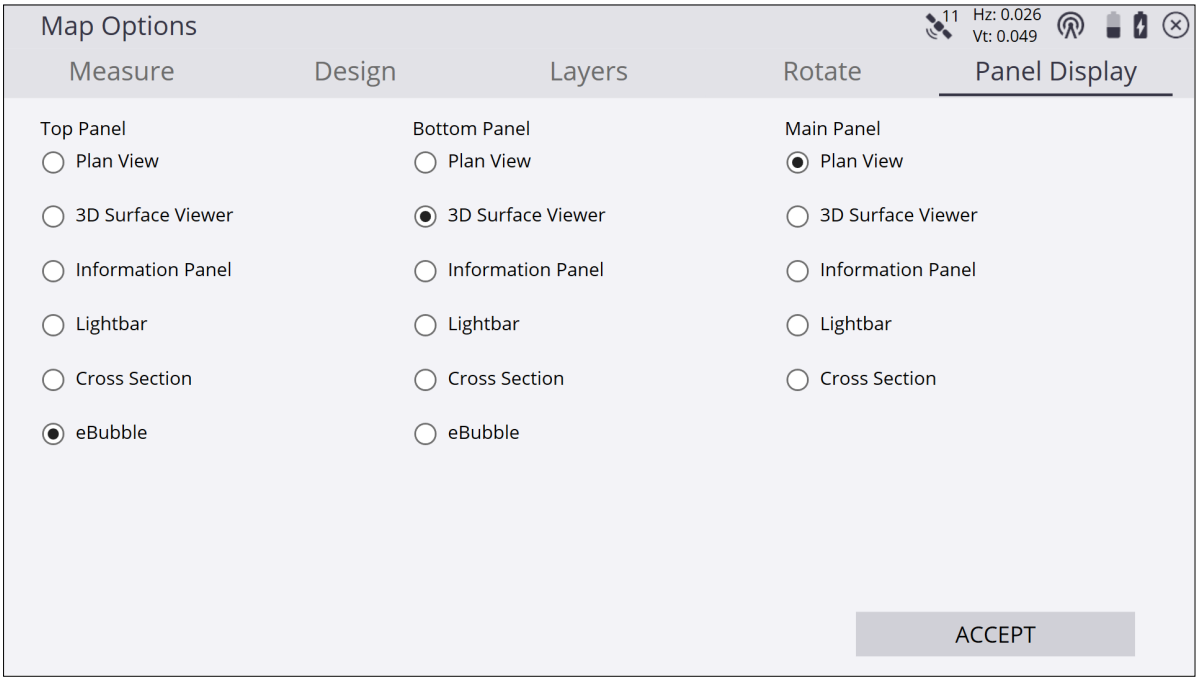
The eBubble is displayed in the top or bottom side panel. It is turned on by tapping on the down arrow and selecting eBubble. .



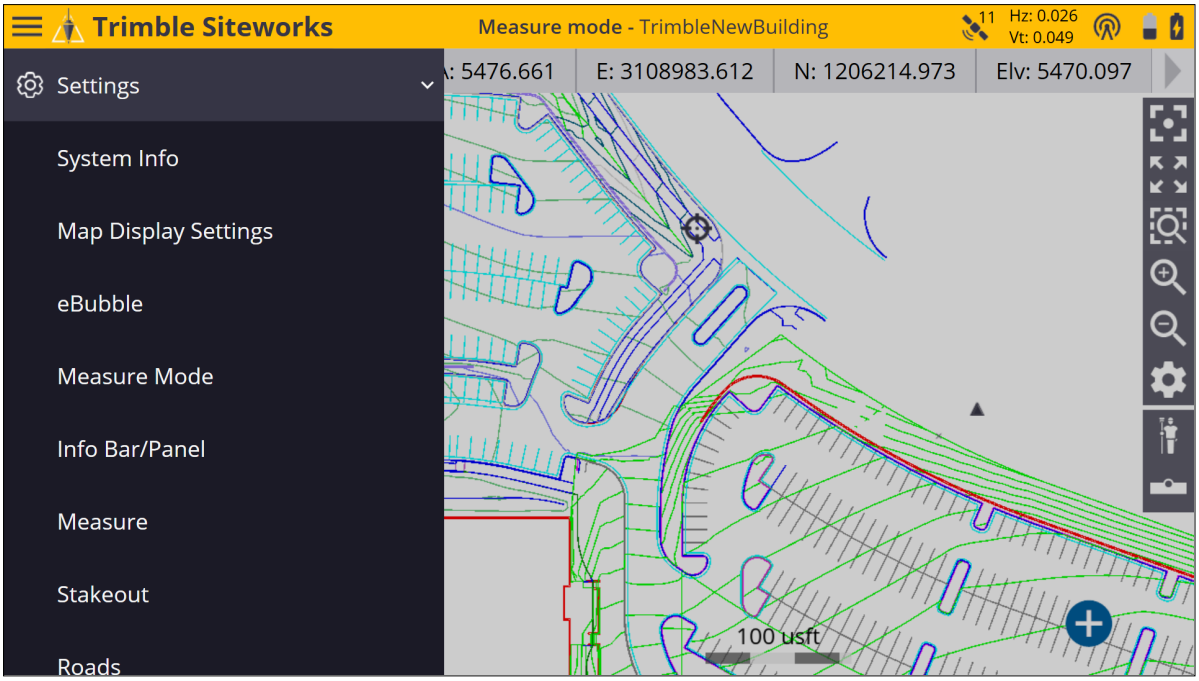
The eBubble is automatically available whenever a compatible receiver is connected to the Siteworks software. It shows the degrees of tilt in the lower left, and the tilt distance in site units (meters, international feet, US survey feet) in the lower right, calculated based on the current rod height. The bubble turns red when the tilt distance exceeds the tilt tolerance value input in the **eBubble Settings** screen. The eBubble can only be displayed in the side panels and not the main working panel.



The eBubble can also be turned on or off by going to the map display options (select the settings Icon) and choosing eBubble under the **Panel Display** tab of the **Map Options** screen. Select the check box to display the eBubble.

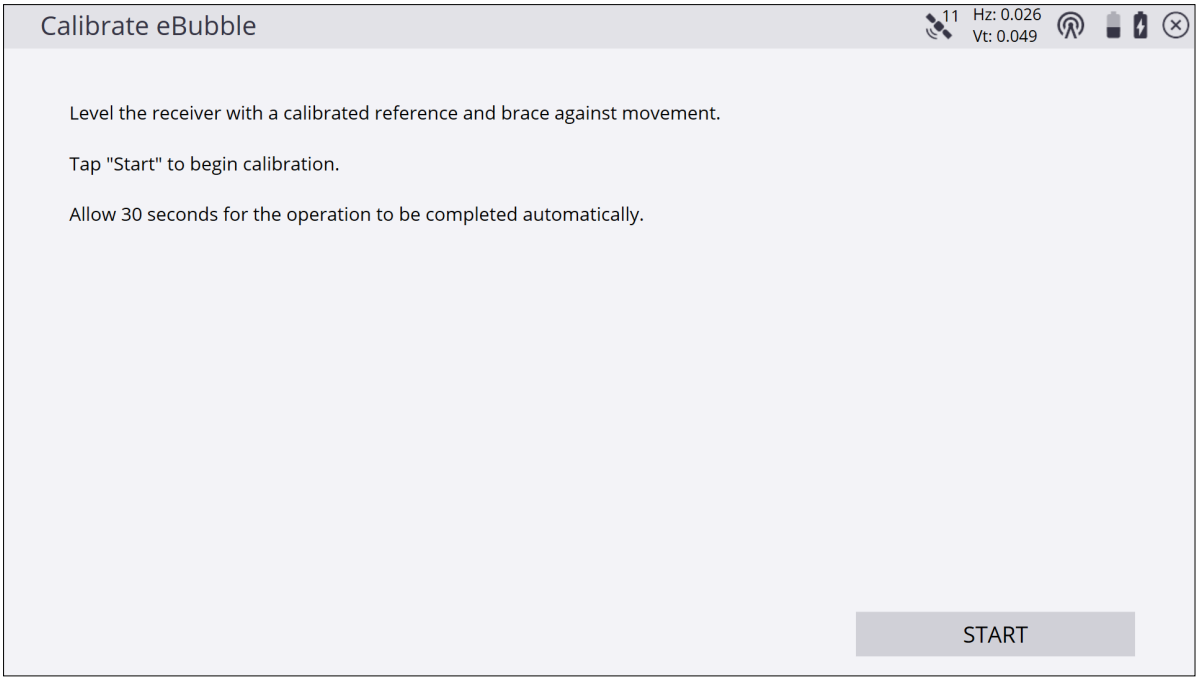


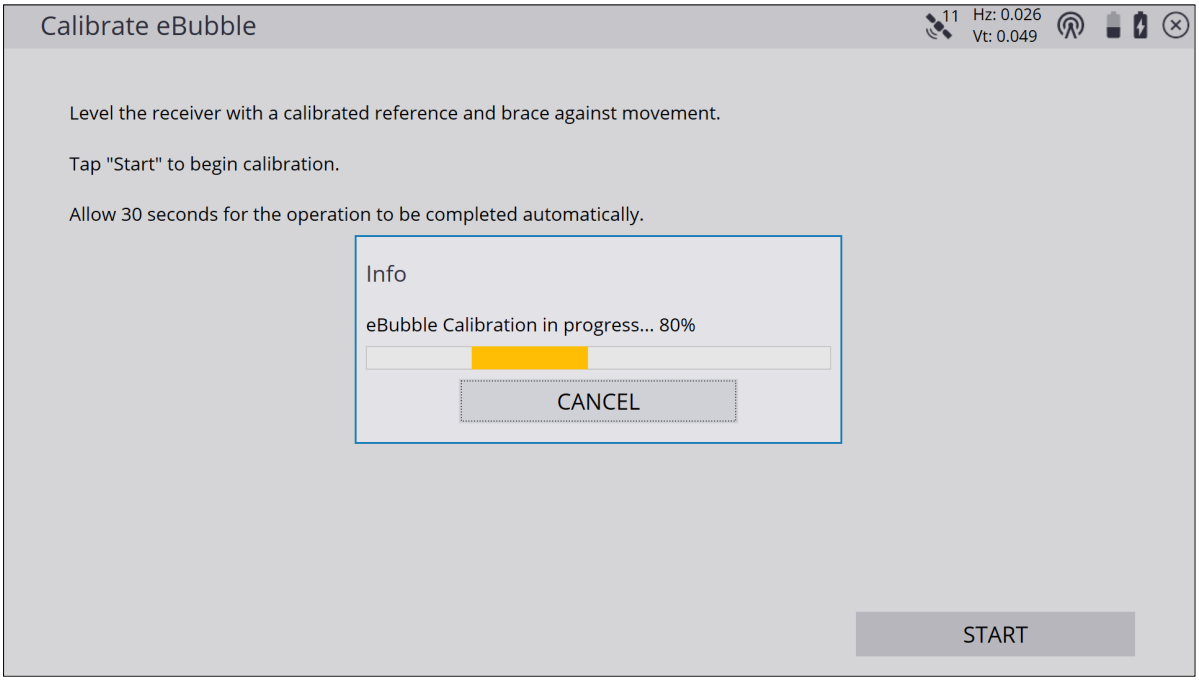
To access the eBubble settings, from the main menu, select **Settings** / **eBubble**.



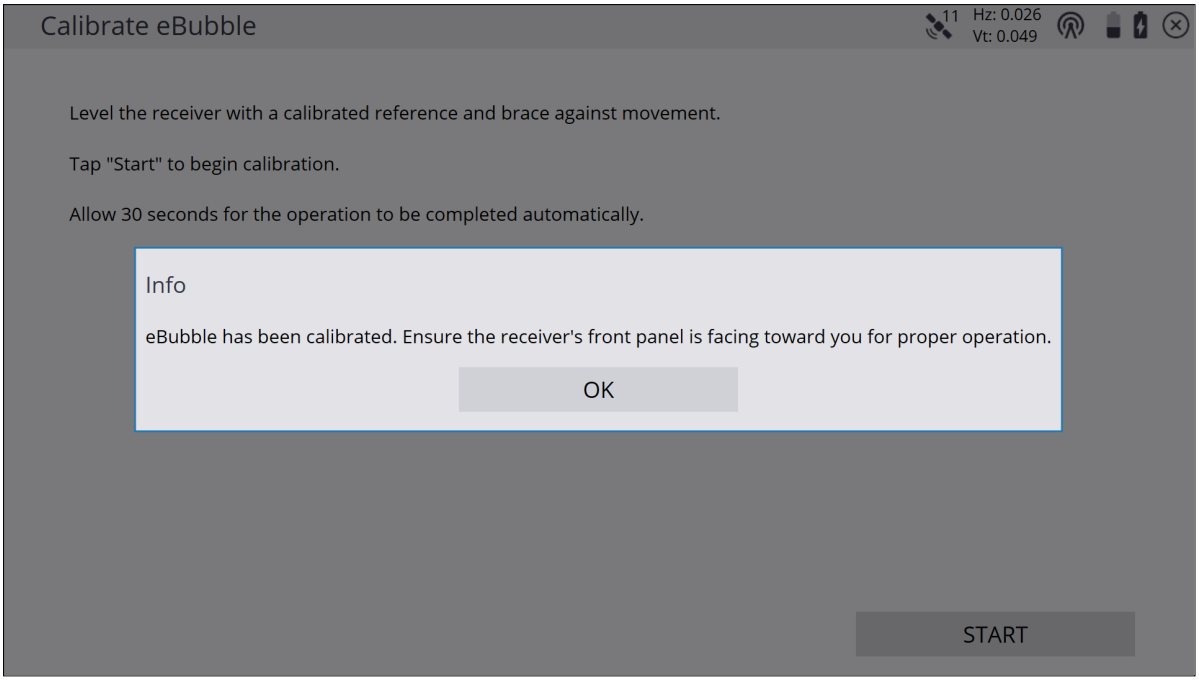
eBubble calibration

The first time a SPS986 smart antenna is used, and every 30 days afterward, it needs to have its tilt sensors calibrated to use the eBubble. To perform a calibration, ensure that the receiver is level and plumb using a calibrated reference (for example on a tripod and calibrated and leveled tribrach) and tap the **Calibrate Tilt Sensor** button to begin the calibration. The receiver must have a clear view of the sky and a satellite lock while performing the calibration, as GNSS time is used to timestamp the calibration. A minimum of four satellites is necessary to be tracked during the calibration. If the receiver is moved significantly during calibration, the process will restart until it is completed without any motion or you cancel it.





After the calibration is complete, a message appears indicating the finished calibration. The receiver's front panel must be facing you for correct operation, as the eBubble's movements and display within Siteworks are oriented based on the receiver's front panel facing the operator. When mounting and using the receiver in vehicle mode, the front panel should be oriented towards the rear of the vehicle .

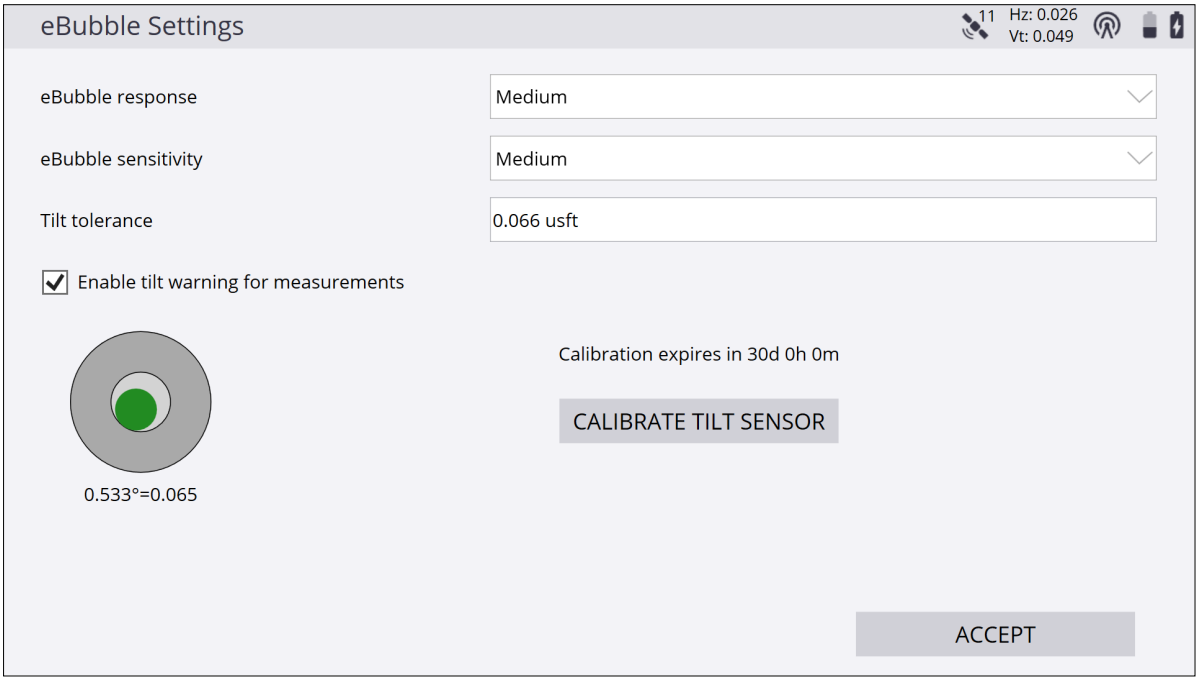


⚠ CAUTION – A well-calibrated eBubble is of vital importance. The accuracy of tilt information used to display the eBubble, and stored with measured points, is totally dependent on the calibration of the tilt sensors inside the GNSS receiver. Using a poorly-calibrated eBubble directly degrades the accuracy of the coordinates measured using the eBubble as a level reference. *Great care should be taken when you are calibrating the eBubble to ensure the most accurate tilt information is available at all times.*

- Bubble reference: Calibrate the eBubble against a properly calibrated physical bubble. The accuracy of the eBubble depends totally on the accuracy of the physical bubble used to calibrate it.
- Pole stability: When calibrating the eBubble, the pole the GNSS receiver is on should be as vertical and as stable as possible. In practice this means using at least a bipod to hold the pole as still as possible.
- Pole straightness: A bent pole affects the tilt measured by the sensors in the GNSS receiver. If you calibrate the eBubble using a bent pole and then change poles, the accuracy of points are affected. Likewise, if you calibrate using a straight pole and then change to a bent pole, the GNSS will not be plumb even though the eBubble will say it is, again affecting the accuracy of the measured points.
- Abuse: If the GNSS receiver suffers severe abuse such as a pole drop, you should recalibrate the eBubble.

eBubble settings

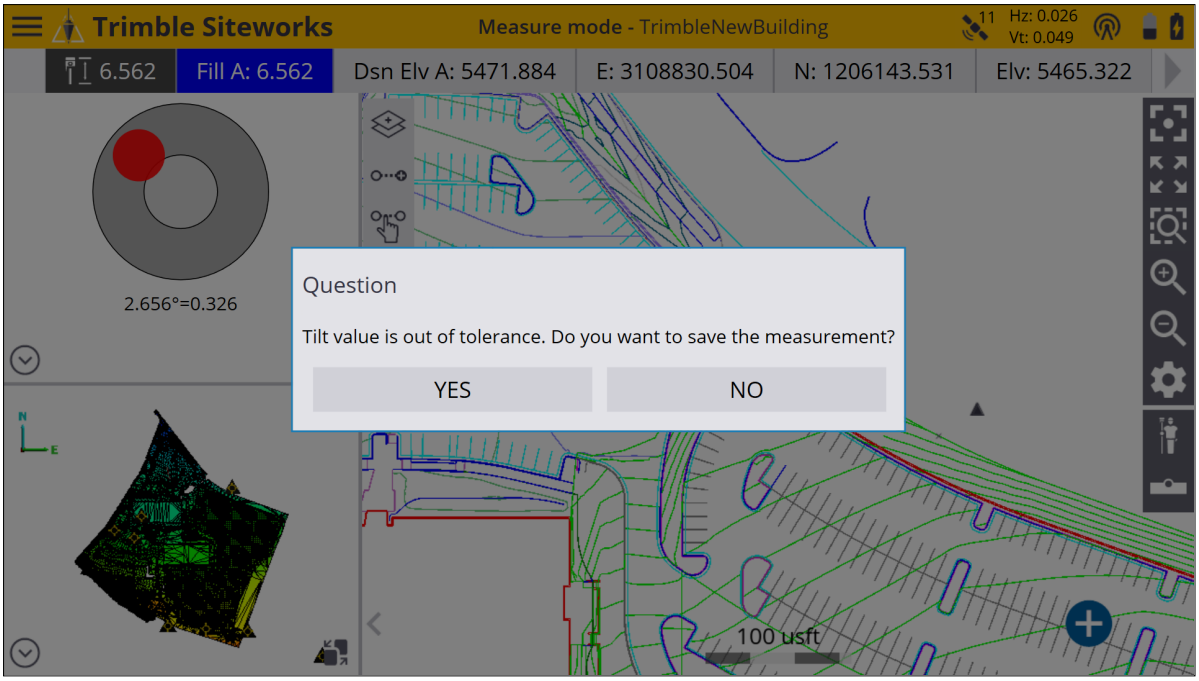
The **eBubble Settings** screen contains controls for the eBubble response, eBubble sensitivity, tilt tolerance, tilt warning, and performing an eBubble calibration:



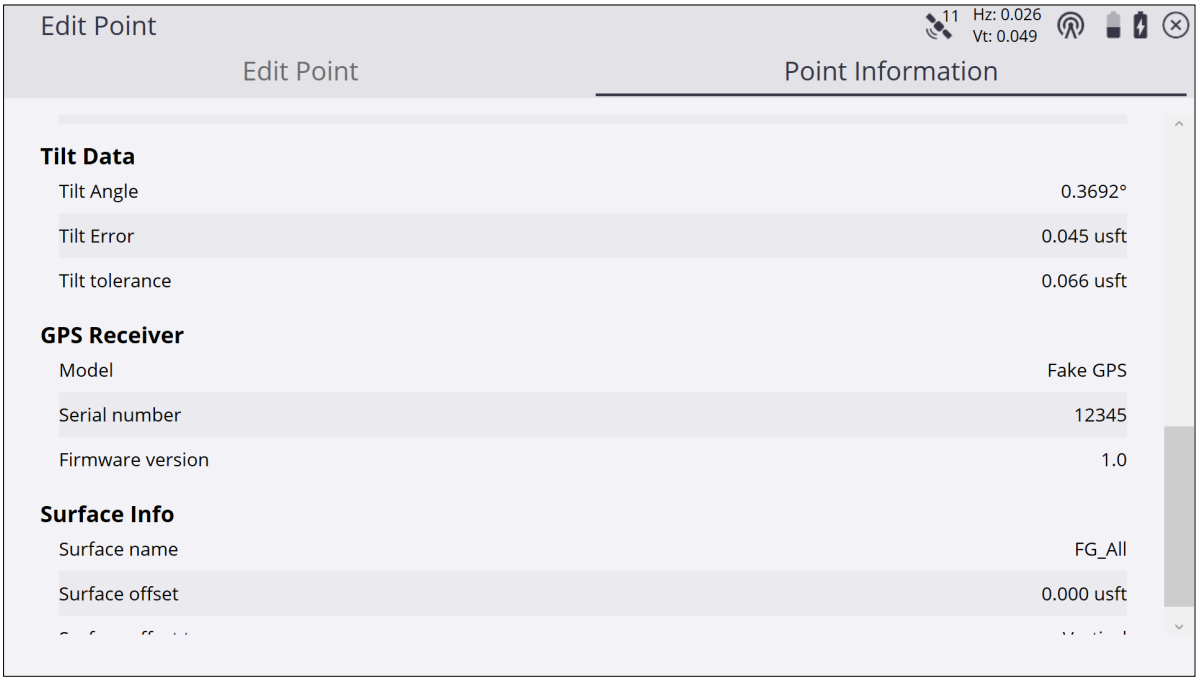
The **eBubble response** setting controls how quickly the bubble's position is updated on the screen, which affects its speed of movement.

The **eBubble sensitivity** setting controls how much the bubble moves on screen for a given change in angle. Select High to make the eBubble move a lot for a given change in angle, and Low to move the eBubble a little for a given change in angle. Changing this sensitivity value also affects the diameter of the Tilt Tolerance display of the inner ring of the eBubble display. The inner ring of the eBubble display represents the tilt value where the calculated tilt distance will exceed the Tilt Tolerance amount. The eBubble will turn red when the bubble touches or extends beyond this inner ring. The diameter of the inner ring changes depending on the current rod height, eBubble sensitivity setting, and the Tilt Tolerance value.

The **Tilt tolerance** setting is the value at which the eBubble display will turn red, and a warning will appear if a measurement is taken while the tilt distance is greater than the Tilt tolerance value. Select the **Enable tilt warning for measurements** check box to activate a warning message if a measurement is made and the tilt distance is greater than the Tilt Tolerance value.



Tilt data are stored with each point measured with a compatible receiver and can be viewed by tapping-and-holding on a point and selecting **Point Information**, or by viewing point information via the Point Manager. The Tilt Angle, Tilt Error, and Tilt Tolerance values are stored. These tilt values are included in exported CSV data if the **Include QA data** option is selected.



11

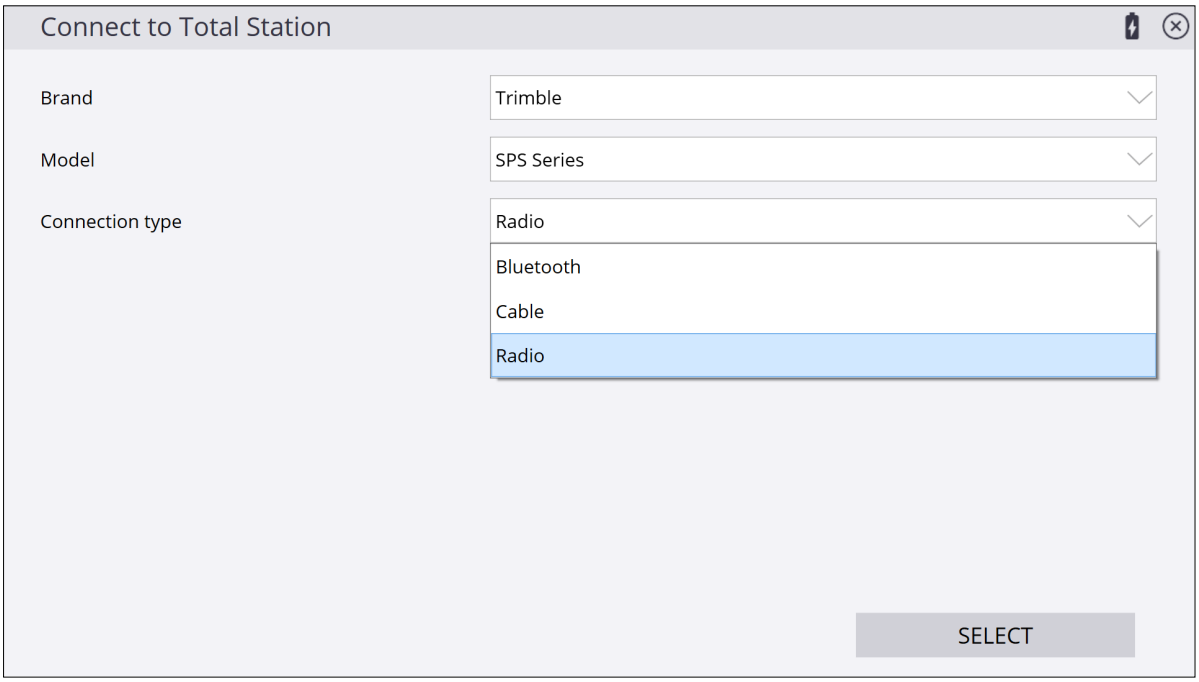
Measuring with a Total Station

To perform measurement or stakeout tasks, you need the Siteworks site controller software running on a controller that is connected to a positioning device. This chapter explains how to set up and use SPS Series total stations with the Siteworks software.

Connecting to a total station

When you start the software the most common connection to the total station is through a radio. The connection can also be made through Bluetooth or a cable although both these methods require you to be in close proximity to the total station. When the connection is made, the **Level Compensator** screen (see Leveling the total station, page 157) appears.

The connect to total station screen either automatically appears during the project startup process or can be accessed via **Home** menu / **Project Setup** / **Connect Device**and selecting total station. This opens the **Connect to Total station** menu where you can select the connection type.



If you are setting up a radio connection for the first time:

1. Select Trimble as a Brand.
2. Select the model of total station you are using. The most common is the SPS series.
3. Select Radio as the connection type.
4. Ensure the total station is turned on and configured for a radio connection. The screen of the total station should say waiting for connection.
5. Select the Radio channel and Network ID that are displayed on the total station screen.
6. Tap **ACCEPT** and follow the instructions that appear on the screen.

If you are using Bluetooth wireless technology to connect to the total station:

1. Select Trimble as a Brand.
2. Select the model of total station you are using. The most common is the SPS series.
3. Select Bluetooth as the connection type.
4. Ensure the total station is on and the Bluetooth is turned on in the total station. The screen of the total station should say waiting for connection.
5. Enter the PIN code set in the total station.
6. Tap ACCEPT and follow the instructions that appear on the screen.

If you leave the controller in GPS mode, it cannot find the connection even if it is connected directly to a total station.

To connect to the instrument via cable:

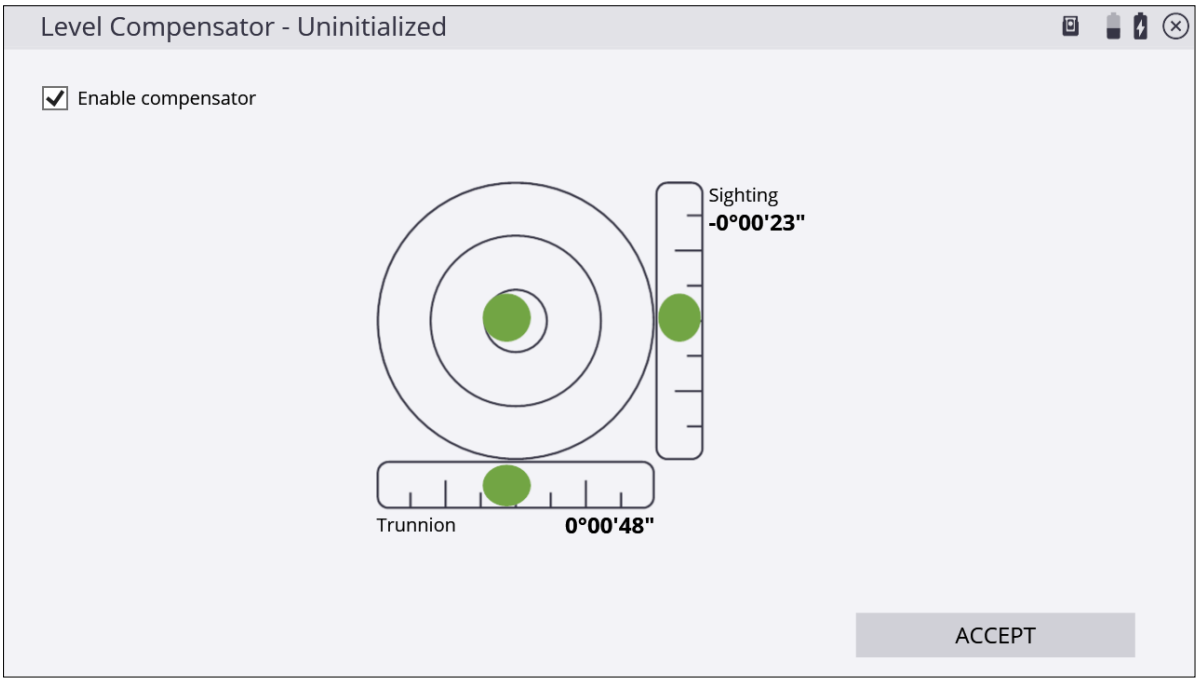
1. Select Trimble as a Brand.
2. Select the model of total station you are using. The most common is the SPS series.
3. Select Cable as the connection type.
4. Tap ACCEPT and follow the instructions that appear on the screen.

Disconnecting from a total station

After successfully connecting to the total station, a disconnect button appears in the Home menu/ Project Setup. Tapping this button disconnects the controller from the total station and places the total station into standby mode, where it can be picked up by a GCS900 or Earthworks machine control system, or re-connected to in Siteworks. This feature is beneficial if you are using machine control systems, as you no longer need to travel back to the instrument to physically power cycle the instrument or unplug the battery to put the instrument into standby mode so a GCS900 or Earthworks system can connect.

Leveling the total station

The compensator inside the total station provides dual-axis correction for any misleveling of the instrument during operation, for a working range of up to 6' of arc. The display enables you to level the total station accurately using the foot-screws on the tribrach. Note if the total station is too far off level, no level data will appear in this screen. To remedy this, get the bubble on the total station tribrach approximately level



To level the total station:

1. Align the front face of the total station with two tribrach foot-screws.
2. Adjust the horizontal bubble in the display using those same two foot screws.
3. Adjust the vertical bubble using the third foot screw.
4. Once level, tap ACCEPT.

You can disable the compensator, however, *only do this in extreme working conditions* such as when working:

- in close proximity to a piling rig, vibratory compactor, or other source of extreme ground vibration that would continuously affect the compensation effect on the total station.
- on a mobile platform where the level is continuously changing, but where all measurements are needed in the reference frame of the platform itself, for example on a marine vessel or oil platform.

- in extremely windy conditions where wind buffet on the total station is significant, and would continuously affect the compensation of the total station.

If the compensator is switched off, the total station stops correcting for instrument mislevel. In these situations, watch and adjust the level on a regular basis. At this point, Trimble assumes that the total station is set up accurately over a known point (see Setting up on a known point, page 165) or that you have set it up in a convenient location, and will establish its position via a resection using the Arbitrary location (see Arbitrary location, page 159).

Establishing the station

You must know the total station position and orientation if the total station is to be used for site measurement or stakeout operations where the positions computed are related to a project coordinate system. There are two methods of establishing the position and orientation of the total station:

- Arbitrary location (see Arbitrary location, page 159, *also known as free station or resection*)
- Setting up on a known point (see Setting up on a known point, page 165)

Once the station setup has been established, the total station can be used for measurement or stakeout operations.

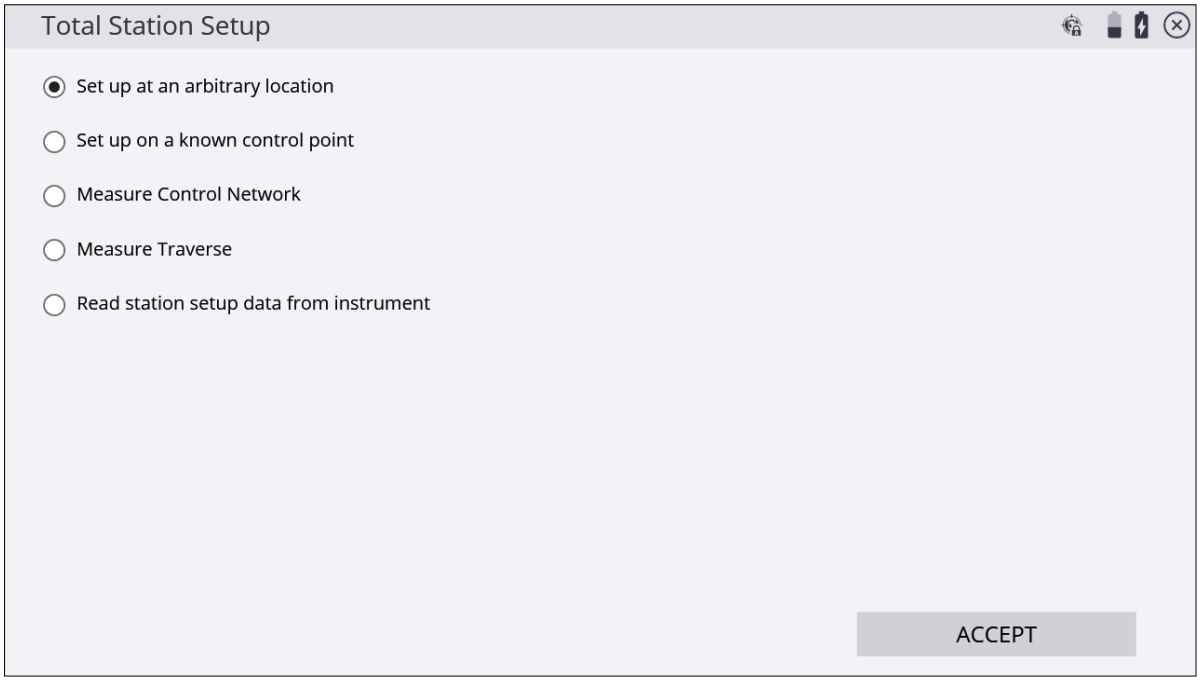
If you have not yet established the position of the total station and you select a measurement or stakeout function, the software automatically forces you into the instrument setup process when connected to a total station. If you have not entered any control point data for the site, the station establishment process only allows you to set up at a manually entered coordinate location and orientation of the angle system to 0.000 in a chosen direction. This method is acceptable only if you are measuring a new site that has not been measured before and provided that you are not trying to measure or stake anything from a loaded design. In most cases, you should have control points available.

You can take angle-only measurements during a station setup, both when the total station is set up on an arbitrary location or on a known control point. Using the angle-only option will not calculate any distance errors in the setup table. When performing an angle-only setup on an arbitrary location, you must shoot a minimum of three control points. Only two control points are necessary when a distance measurement is taken.

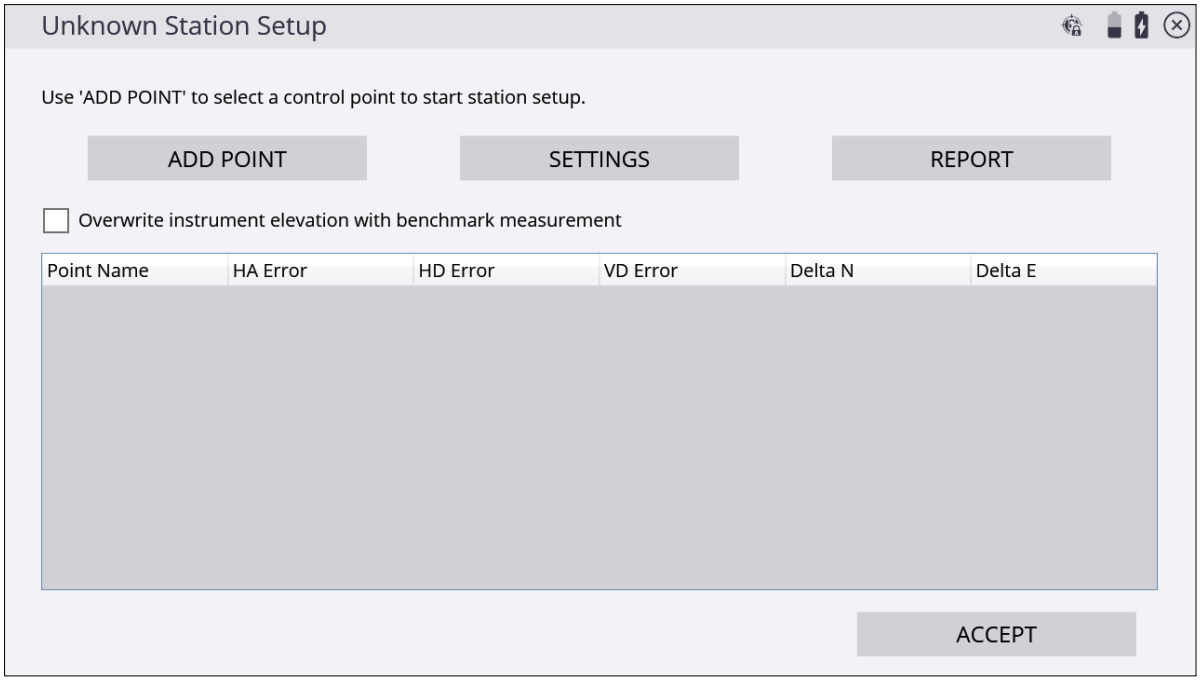
Arbitrary location

The arbitrary location setup enables you to set up the total station at a location that is convenient for the operation to be carried out (and not over a known point). It measures the angle and distance to two or more known points to determine the position and orientation of the total station.

1. In the **Home** menu, select **Project Setup** / **Total Station Setup**. If you just connected to the total station, the software prompts you for the station establishment method. Select the **Set up at an arbitrary location** option:



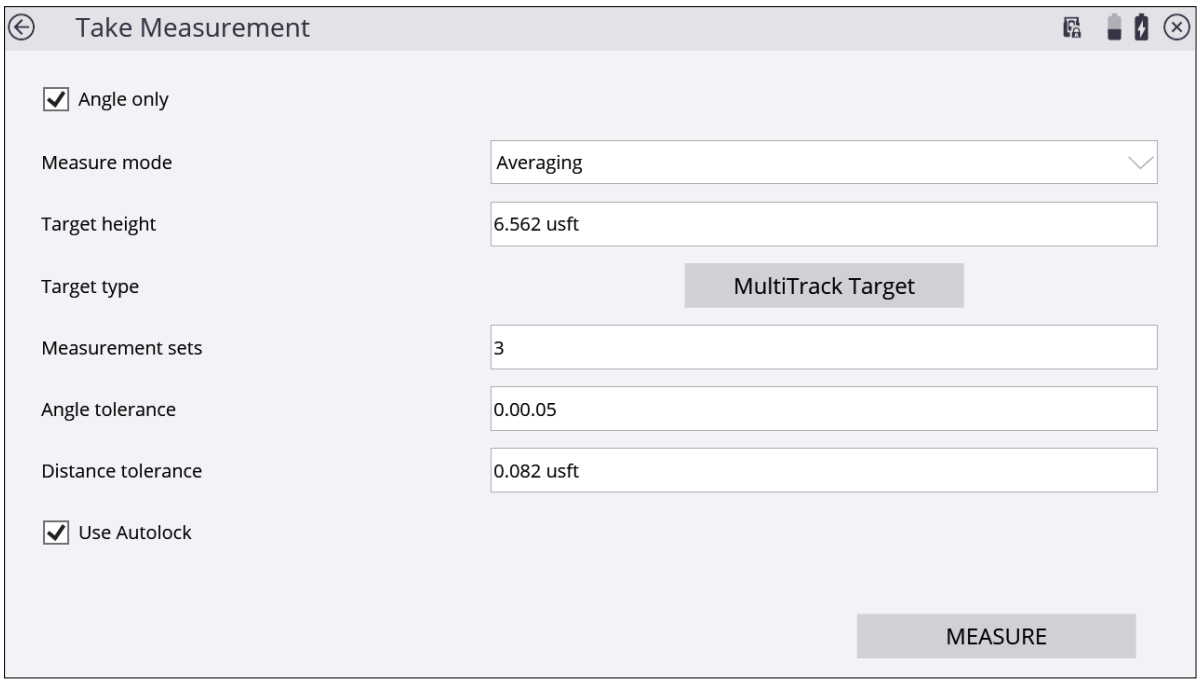
2. Add at least two control points:



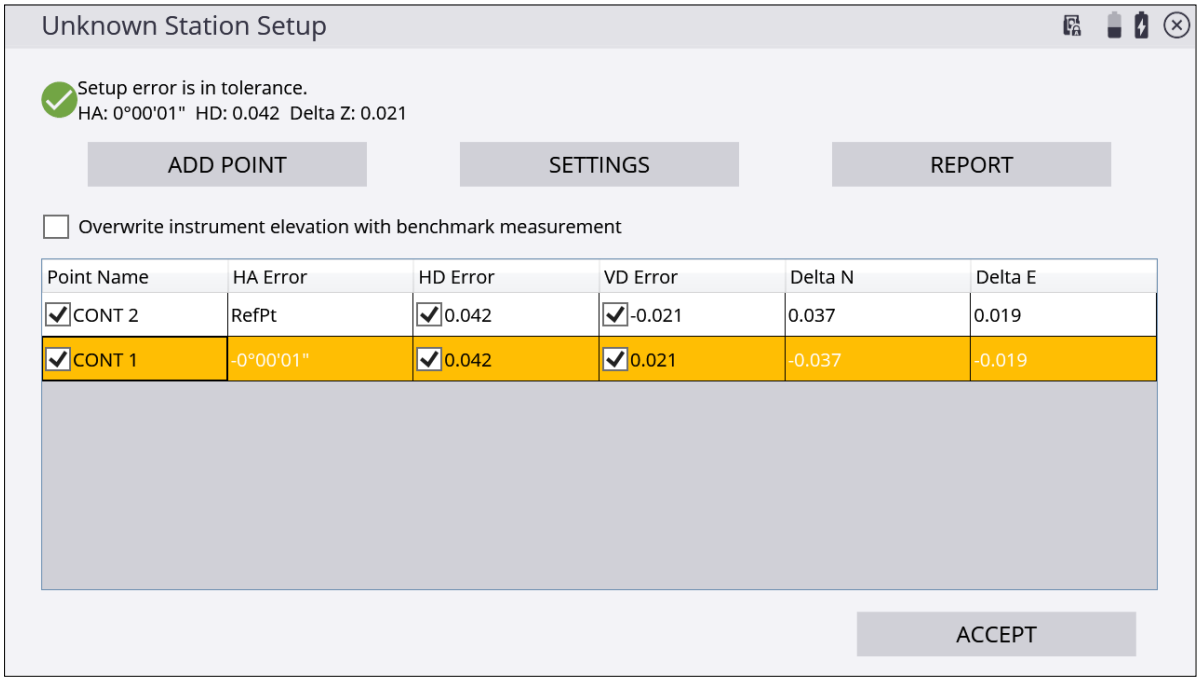
3. Select a point on the map by tapping directly on it and entering the name into the text box, or from the list, by tapping  on the upper right side and then tap ACCEPT:

4. Configure your measurement settings by selecting the measurement mode (Standing, Averaging, DR, DR Target, or DR Averaging), prism target type and height (in Standing and Averaging modes), DR options (in DR, DR Target, or DR Averaging modes),

measurement sets and tolerances (in Averaging and DR Averaging modes), and whether Angle only and/or Autolock (for prism targets) measurements should be used:

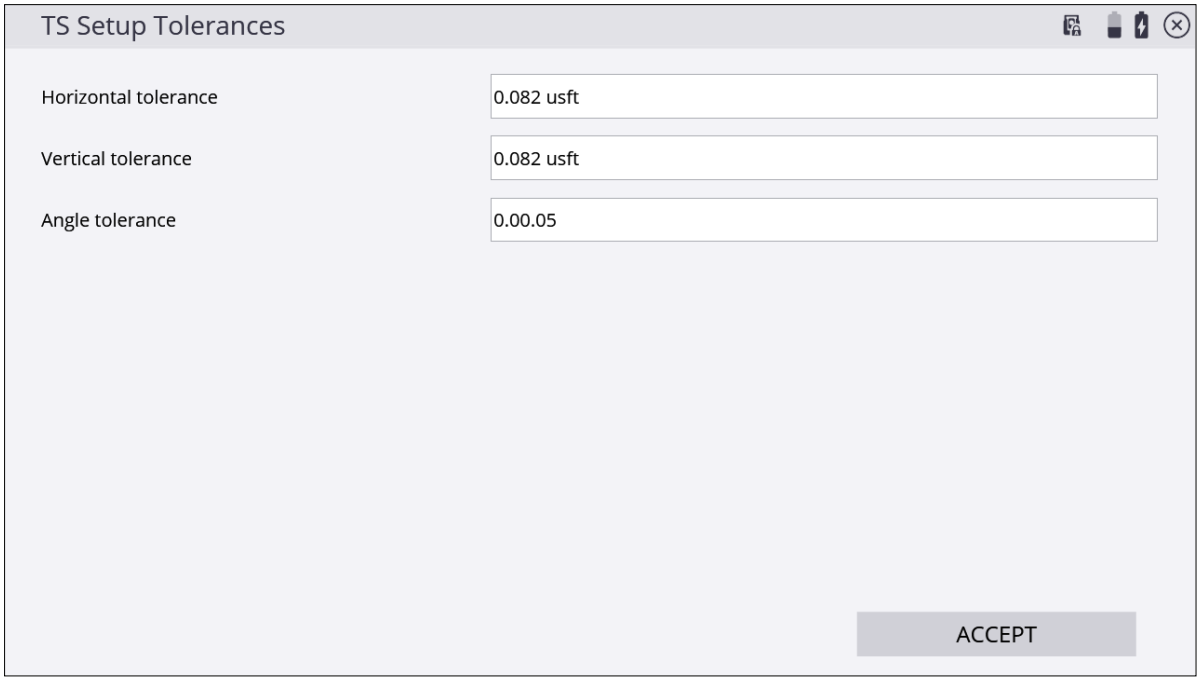


5. After the measurements are complete, the **Station Setup** table will indicate if the setup is in or out of tolerance:

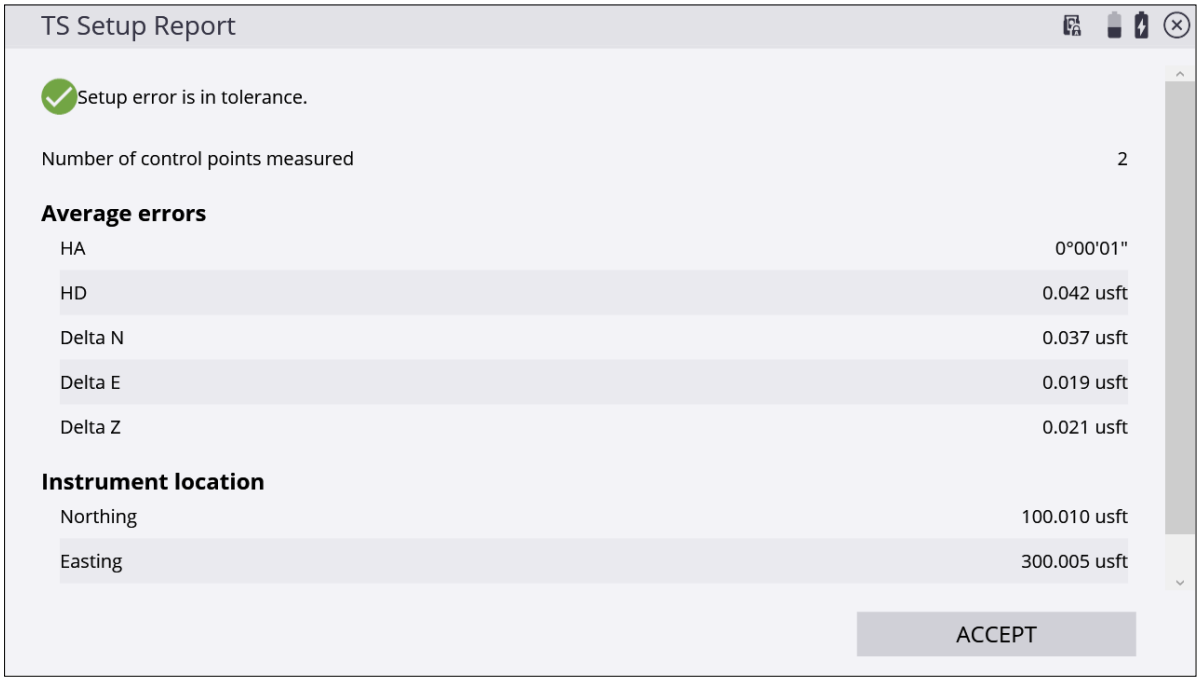


6. Tap **Settings** to adjust the setup tolerances for horizontal and vertical distances and

angle tolerance:

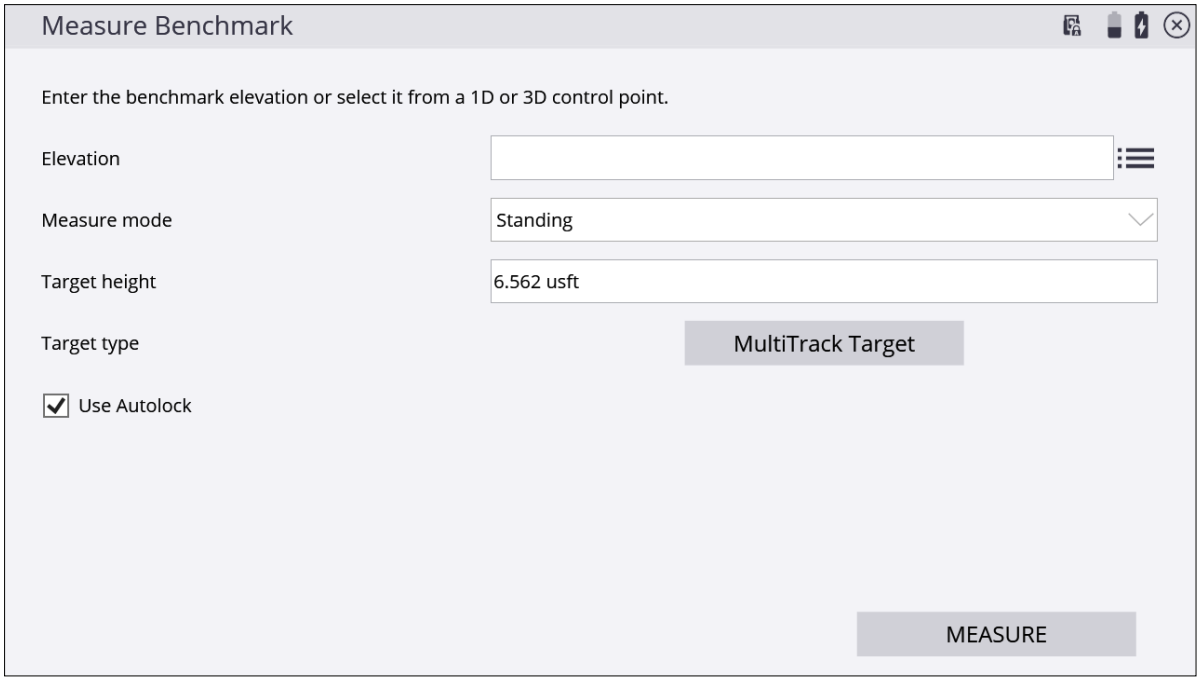


7. Tapping **Report** gives more detailed information about the setup quality of the station:



8. To change the instrument elevation by shooting in a benchmark point, select the **Overwrite instrument elevation with benchmark measurement** check box. At the end of the measurement process, you will be prompted to shoot a benchmark point from

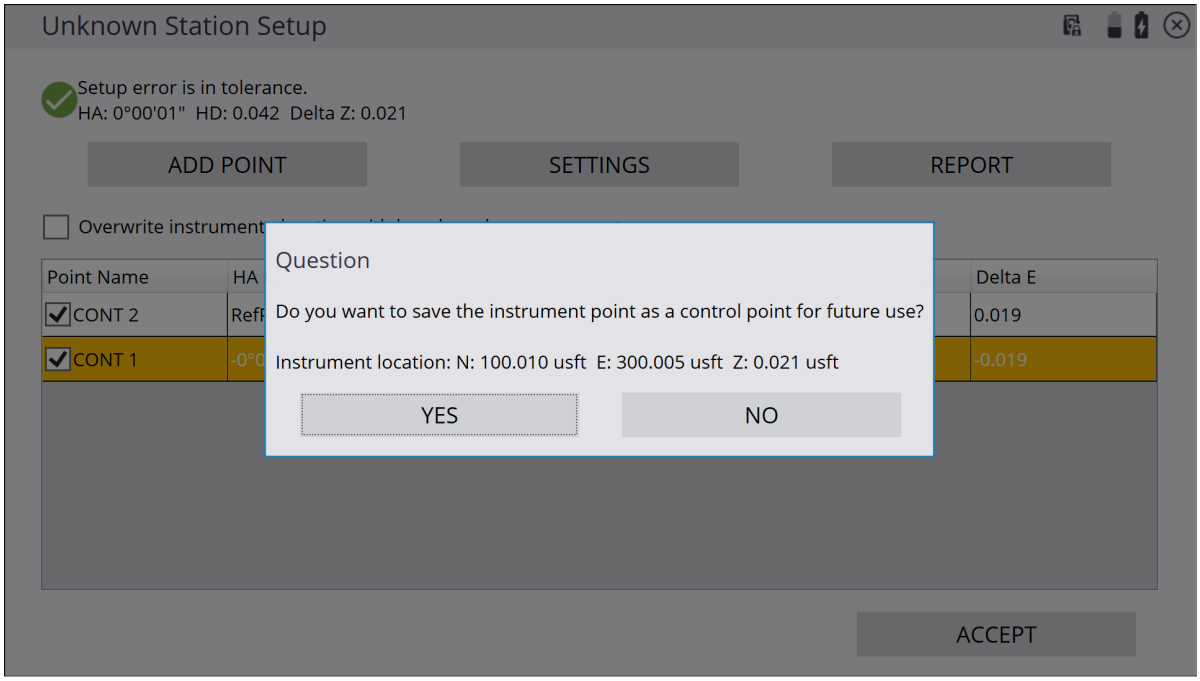
which the station elevation will be calculated:



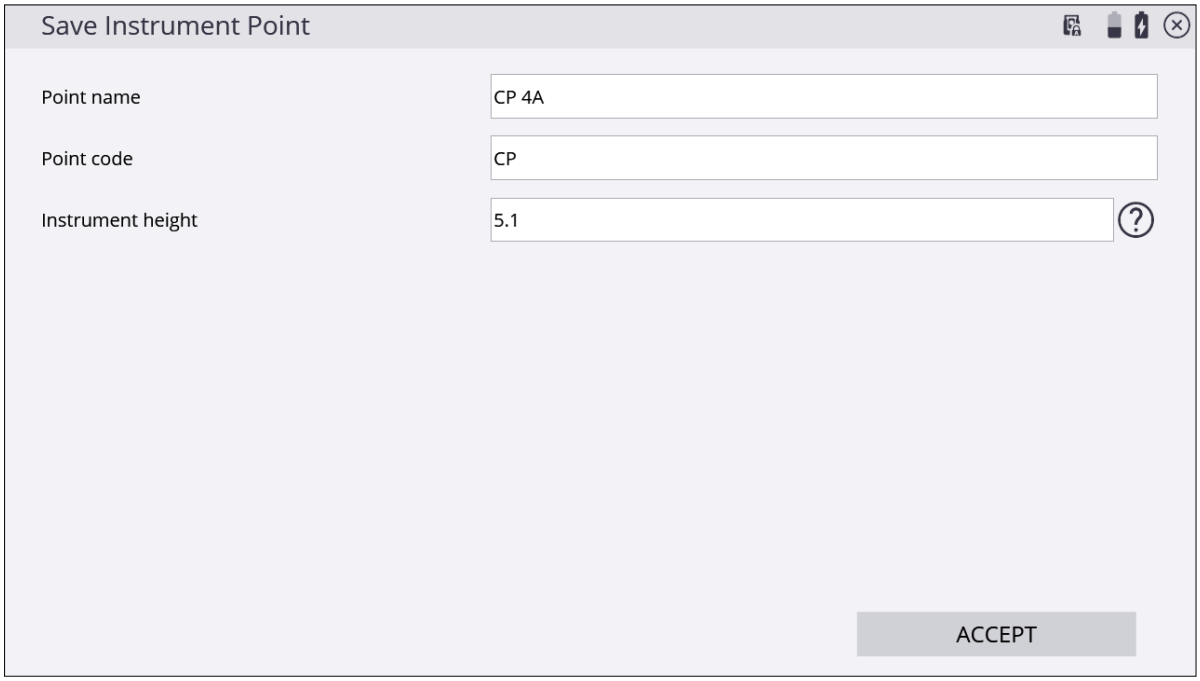
NOTE – The station elevation is calculated relative to this benchmark elevation; it does not simply set the instrument elevation to the benchmark elevation.

9. Tap ACCEPT to complete the arbitrary instrument location setup and view the coordinates of the instrument setup.

   A prompt will appear asking if you would like to save the instrument location as a control point. Tap YES to store the instrument location as a control point, and NO to ignore it:



If you tap **YES**, you are prompted to enter a point name and an instrument height:

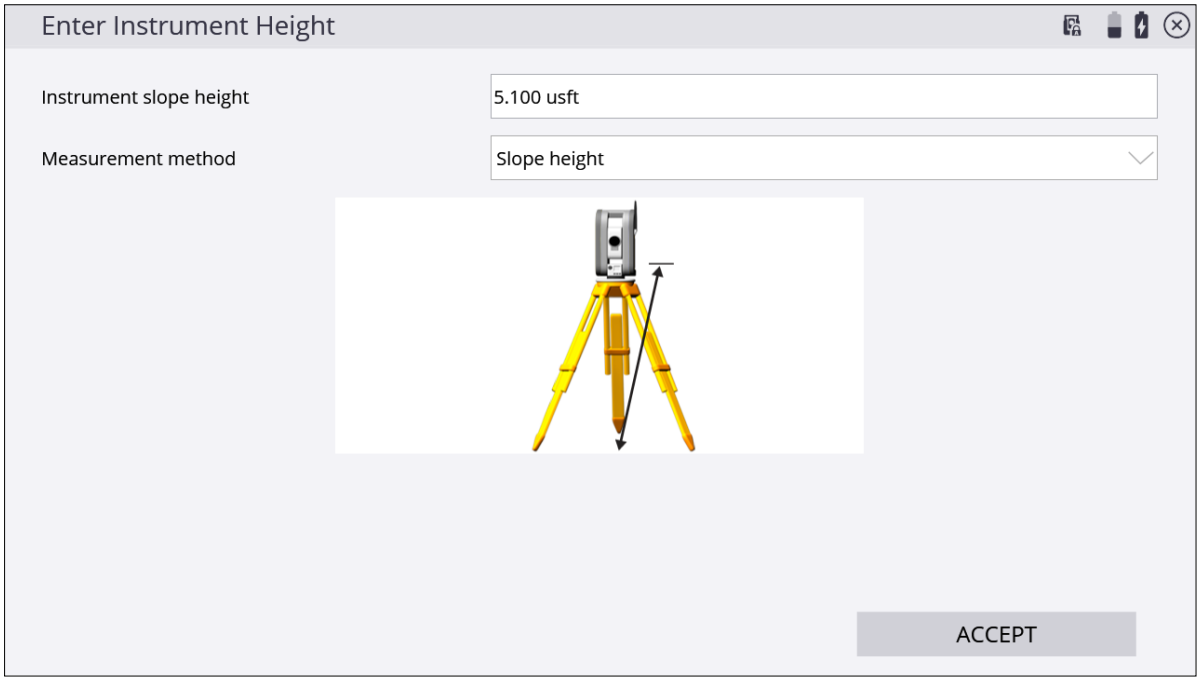


The instrument height can be measured in two ways:

- a vertical height from the ground surface to the center crosshair mark on the side of the instrument

- the slope height, measured directly from the control point to the bottom notch on the side of the instrument.

Selecting the slope height is generally easier to perform more accurately in the field, due to potential height variations between the control point hub and the surrounding ground surface and the ability to get a tape measure to the notch. Tapping the on the Instrument height field opens the following screen, where you can select the measurement method:



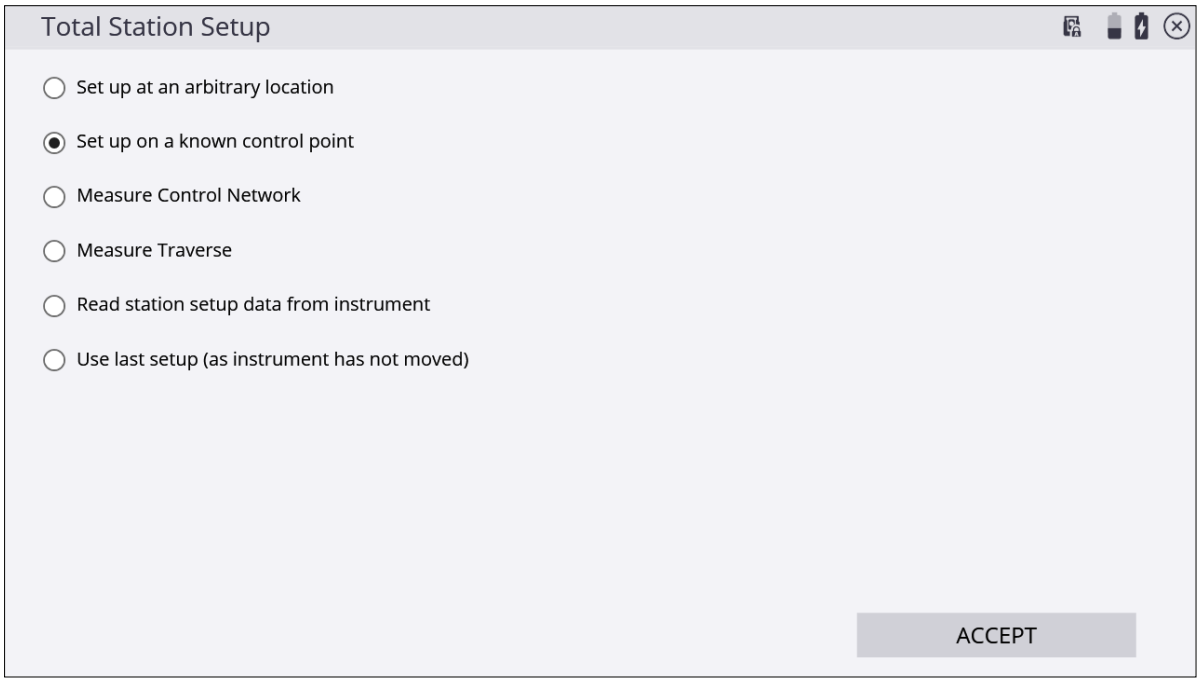
NOTE – If a slope height is entered, the vertical height is automatically calculated by adding .156 m or .511 feet to the slope height. The updated vertical height will be displayed as the instrument height in the ***Save Instrument Point*** *window.*

Setting up on a known point

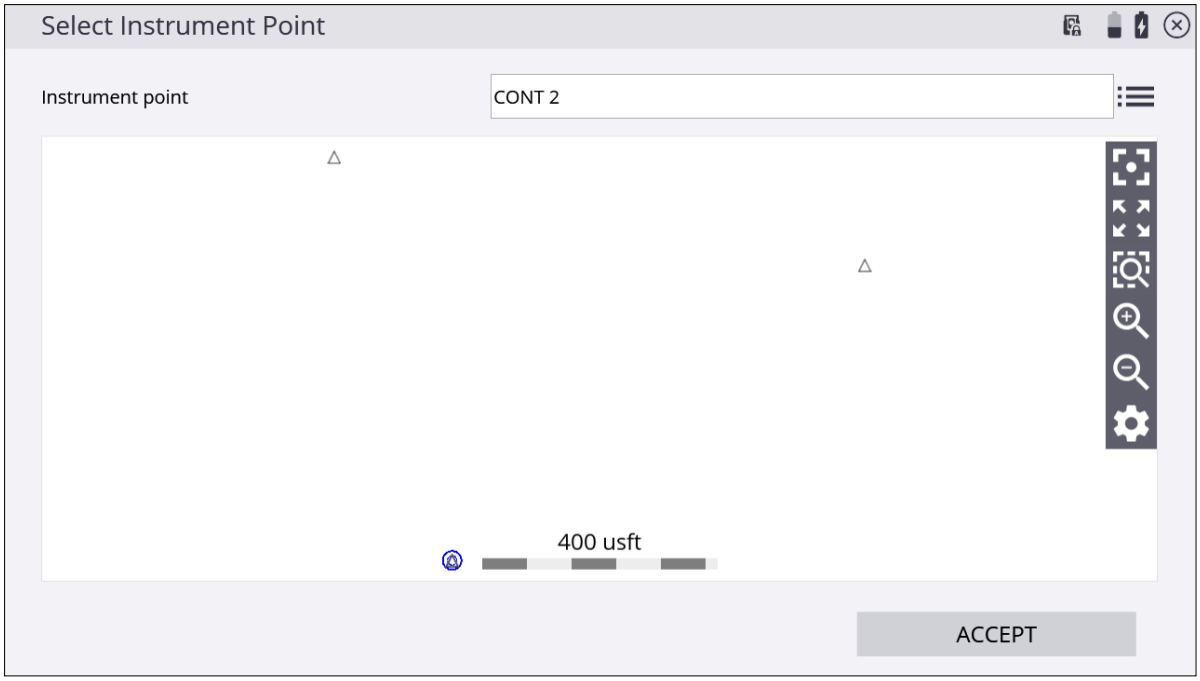
This enables you to set up the total station over a known control point and then measure to one or more reference backsight control points to establish its position and orientation on the jobsite. In most cases, a single reference backsight control point may be all that is required. In other cases, where the accuracy of the work is higher, measuring more than one reference backsight control point can provide better control over the orientation of the total station and a further check that the control point at the total station position or any of the measured reference control point locations has not moved. You can follow similar steps to those used in the arbitrary location method of station establishment. Select from one of the following options:

1. From the **Home** menu, select **Project Setup** / **Connect Device** and tap 

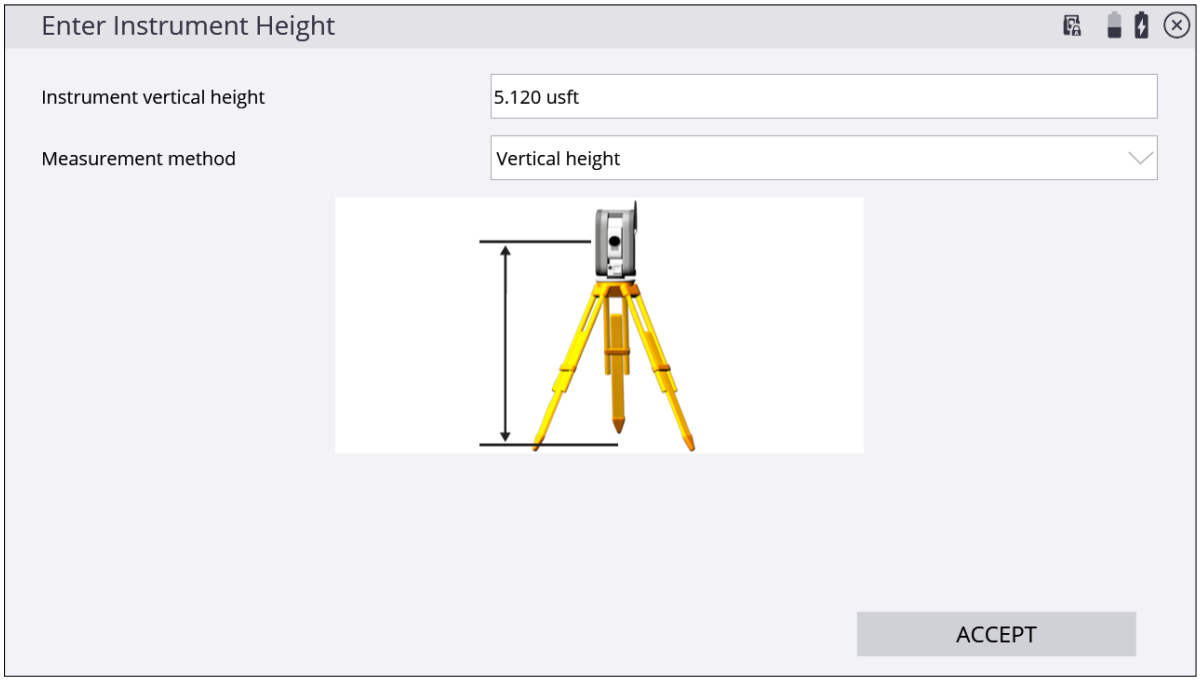
. If you have just connected to the total station, the software prompts you to select the station establishment method. Select the **Set up on a known control point** option and then click **ACCEPT**.



2. Select the instrument's location point by tapping directly on it and entering the name into the text box, or from the list, by tapping  in the upper right side and then tapping **ACCEPT**:



3. Enter the instrument height:



The instrument height can be measured in two ways:

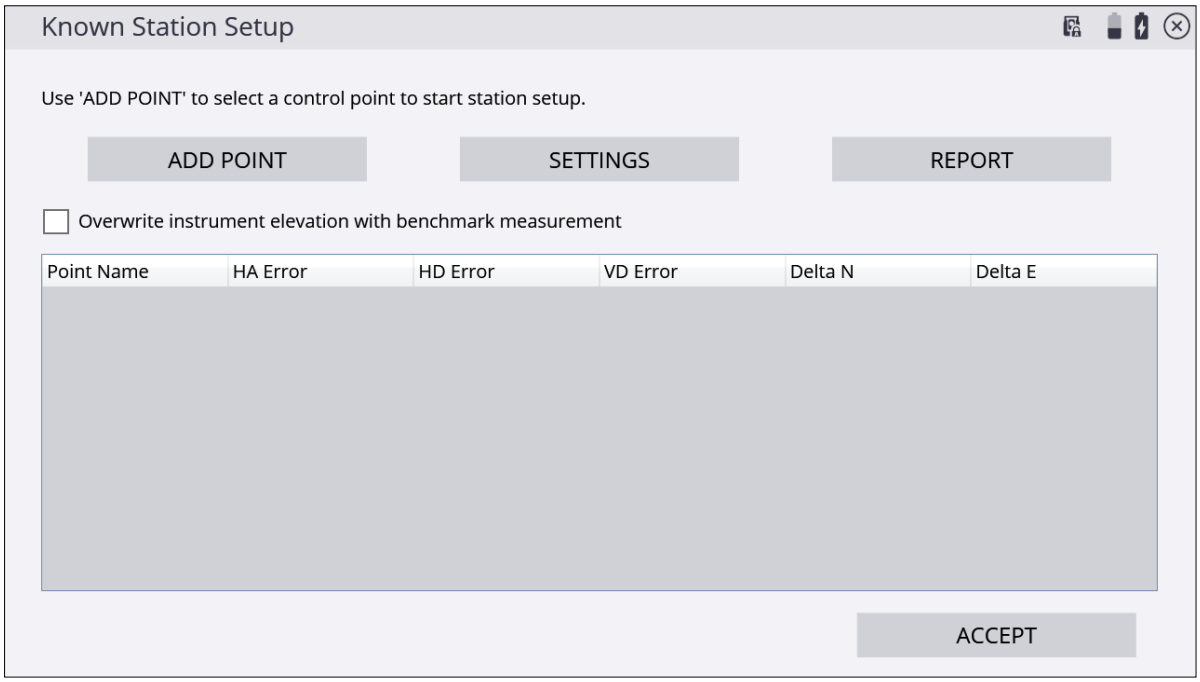
- a vertical height from the ground surface to the center crosshair mark on the side of the instrument

- the slope height, measured directly from the control point to the bottom notch on the side of the instrument.

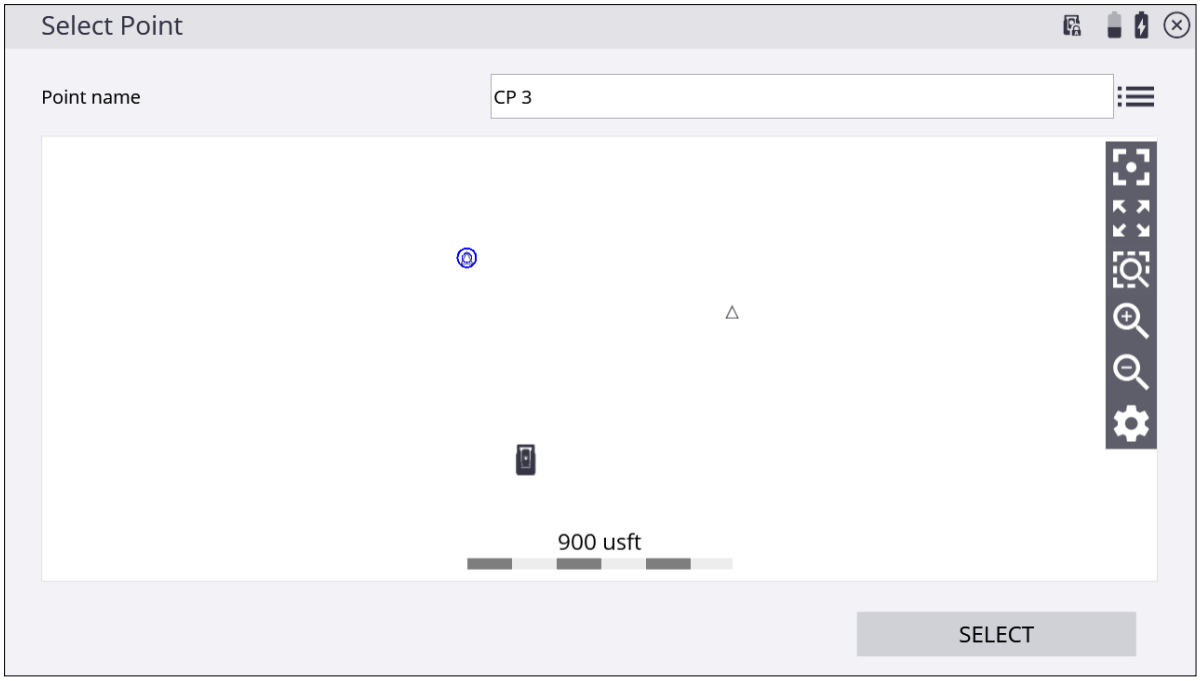
Selecting the slope height is generally easier to perform more accurately in the field, due to potential height variations between the control point hub and the surrounding ground surface.

NOTE – If a slope height is entered, the vertical height is automatically calculated by adding .156 m or .511 feet to the slope height and the updated vertical height will be displayed as the instrument height in the ***Save Instrument Point*** *window.*

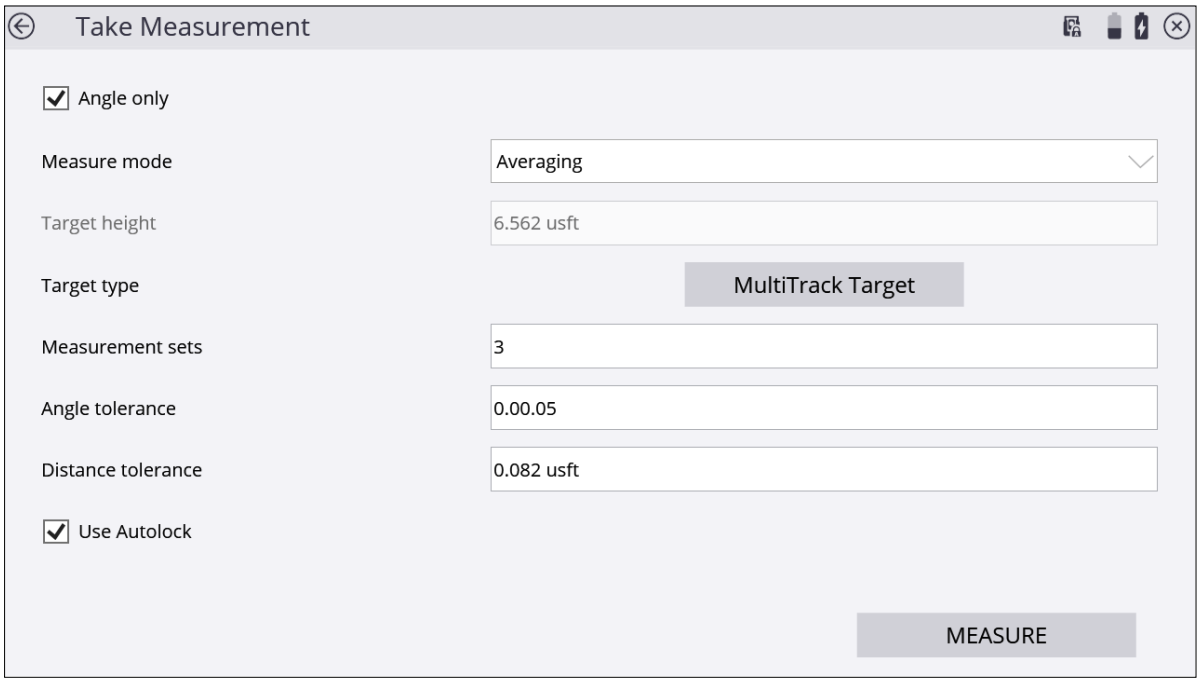
4. Add at least one control point to backsight to by tapping **ADD POINT**.



5. Select a point on the map by tapping directly on it and entering the name into the text box, or from the list, by tapping  in the upper right side and then tapping **SELECT**:

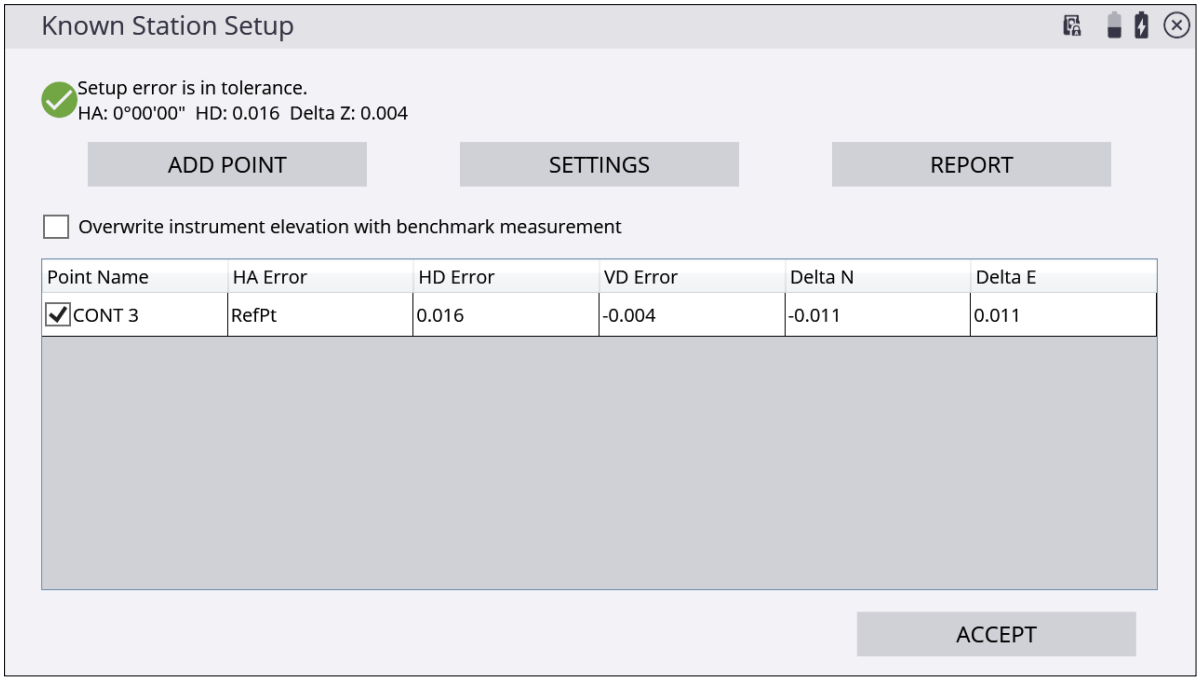


6. Configure your measurement settings by selecting the measurement mode (Standing, Averaging, DR, DR Target, or DR Averaging), prism target type and height (in Standing and Averaging modes), DR options (in DR, DR Target, or DR Averaging modes), measurement sets and tolerances (in Averaging and DR Averaging modes), and whether Angle only and/or Autolock (for prism targets) measurements should be used:

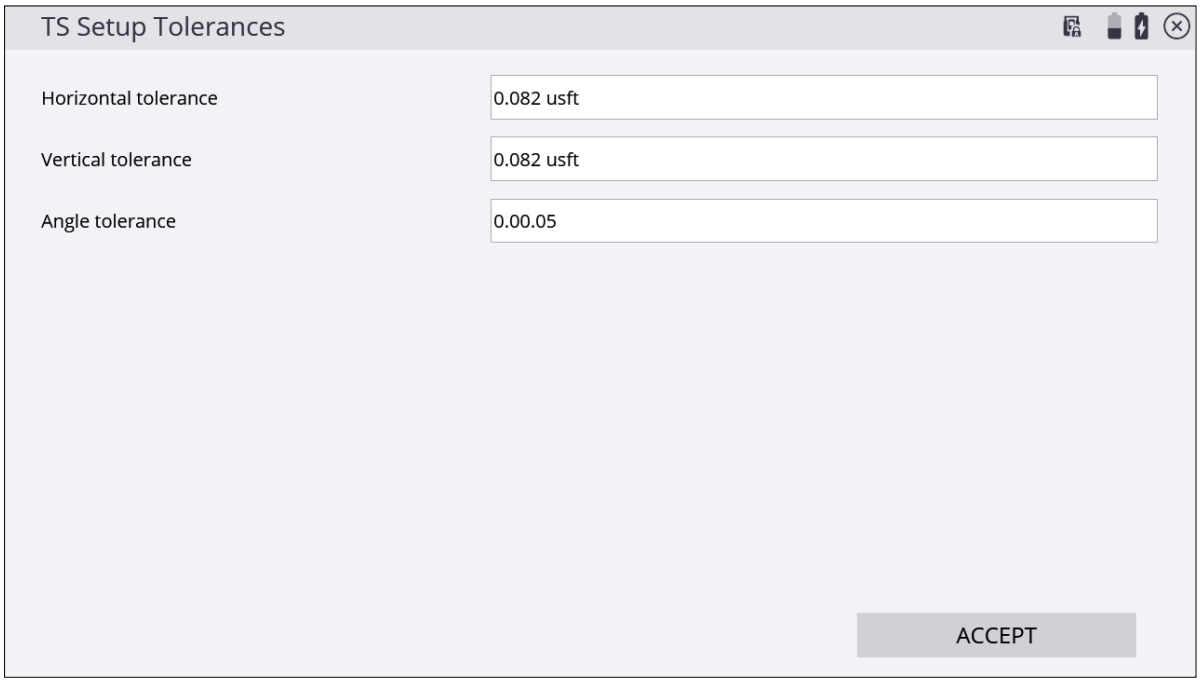


After the measurements are complete, the Station Setup table indicates if the setup is

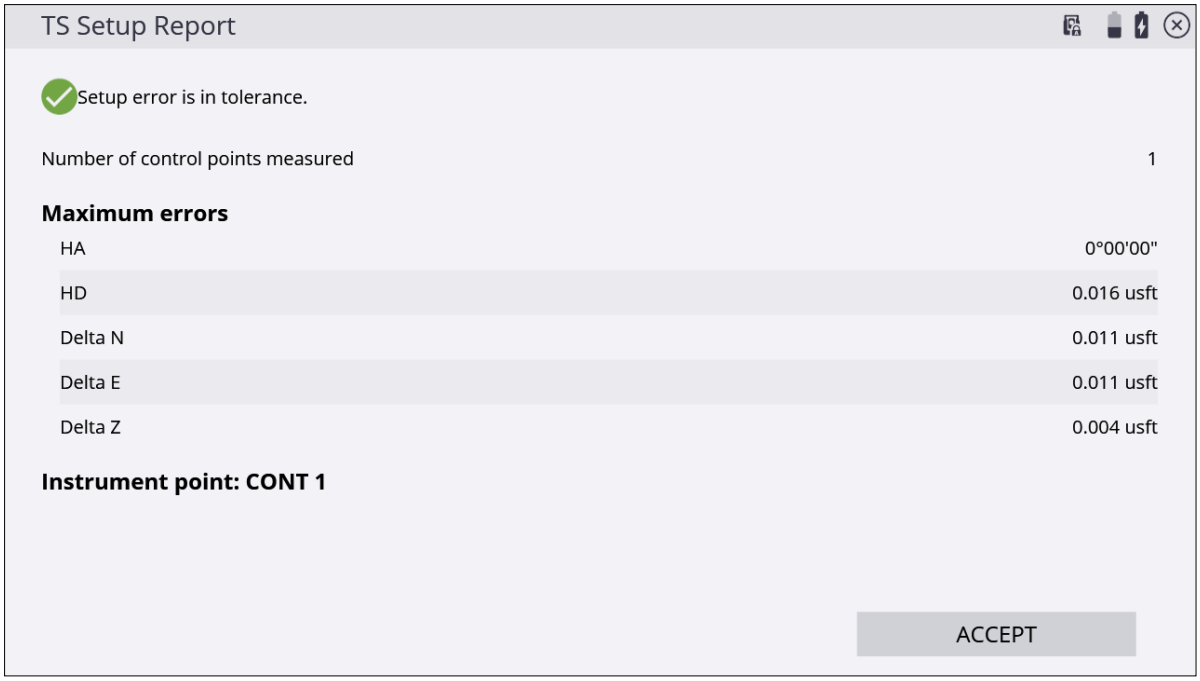
in or out of tolerance:



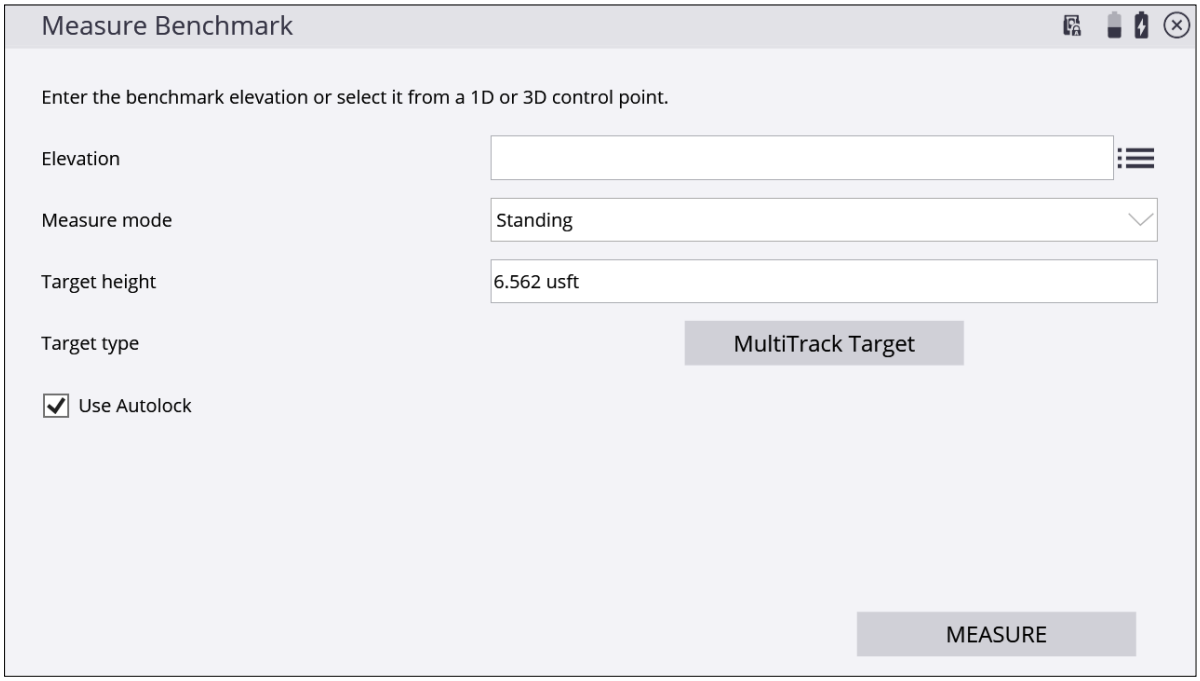
7. Tap **Settings** to adjust the setup tolerances for horizontal and vertical distances and angle tolerance:



8. Tap **Report** to get more detailed information about the setup quality of the station:



9. To change the instrument elevation by shooting in a benchmark point, select the Overwrite instrument elevation with benchmark measurement check box. At the end of the measurement process, you will be prompted to shoot a benchmark point from which the station elevation will be calculated:

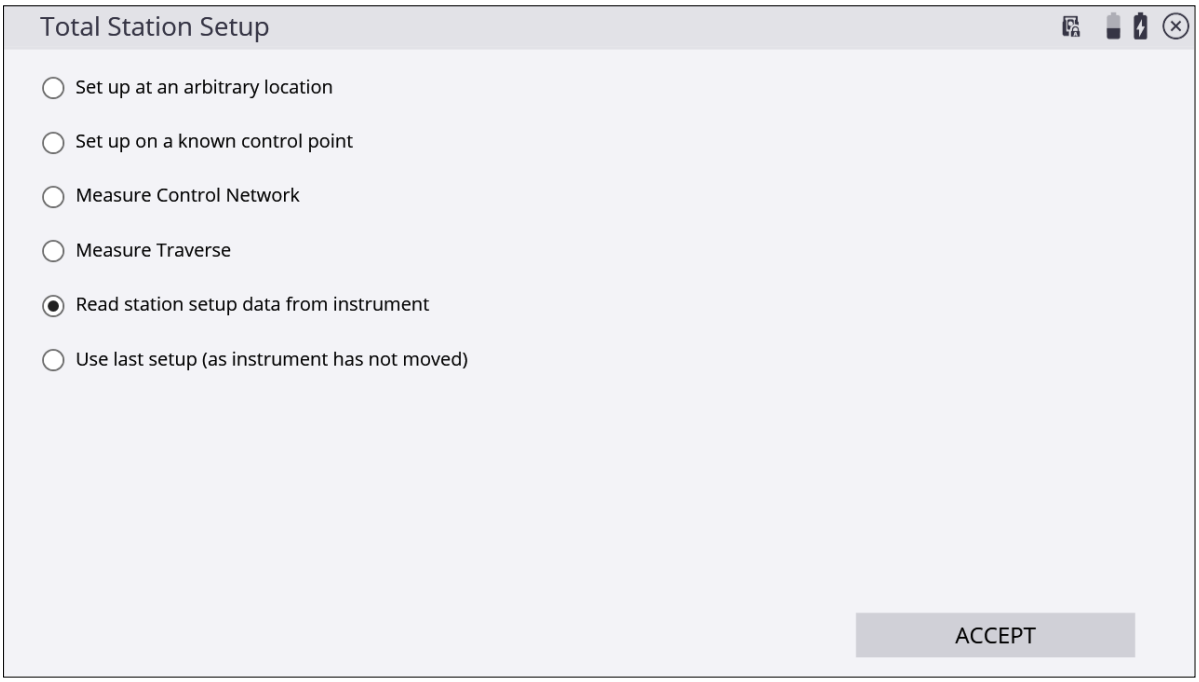


NOTE – The station elevation is calculated relative to this benchmark elevation; it does not simply set the instrument elevation to the benchmark elevation.

Reading station setup data from the total station

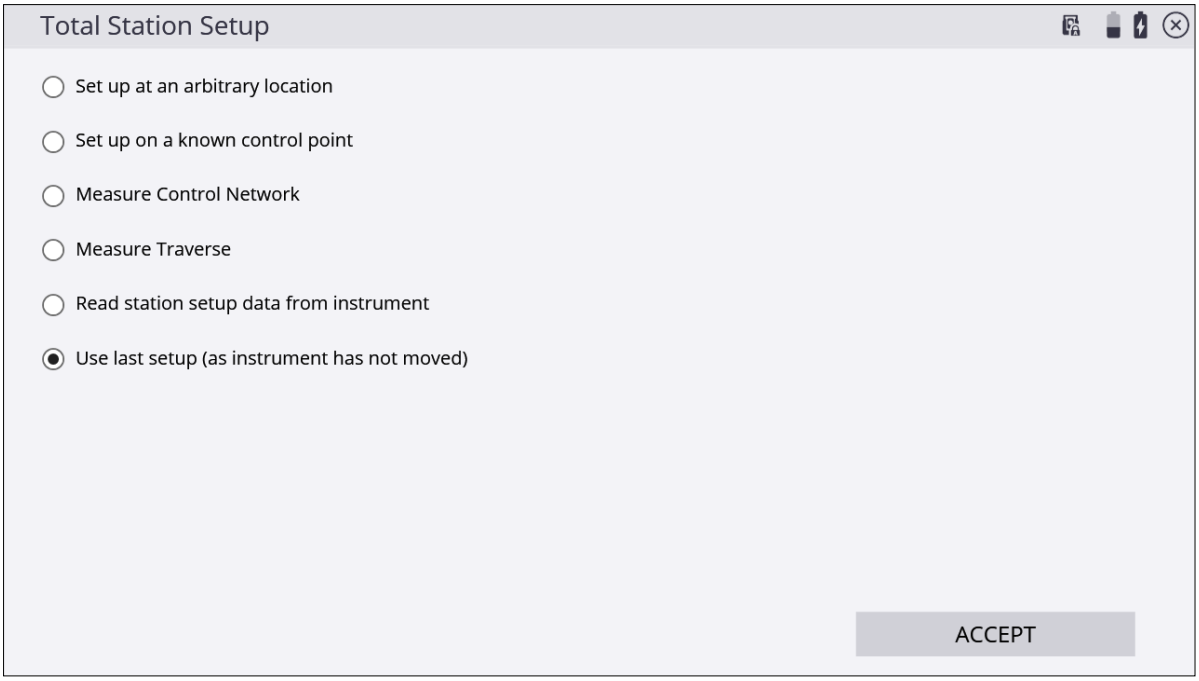
After you have performed a station establishment, the total station stores the information in its memory, so that other controllers running the software can access it. Subsequent Siteworks software users can save time by simply retrieving the station setup information from the total station's memory without resetting up at an arbitrary location or on a known control point.

Note that only one controller is able to connect to a total station at the same time. To retrieve this setup information, connect to the instrument and select Read station setup data from instrument in the Total Station Setup screen:



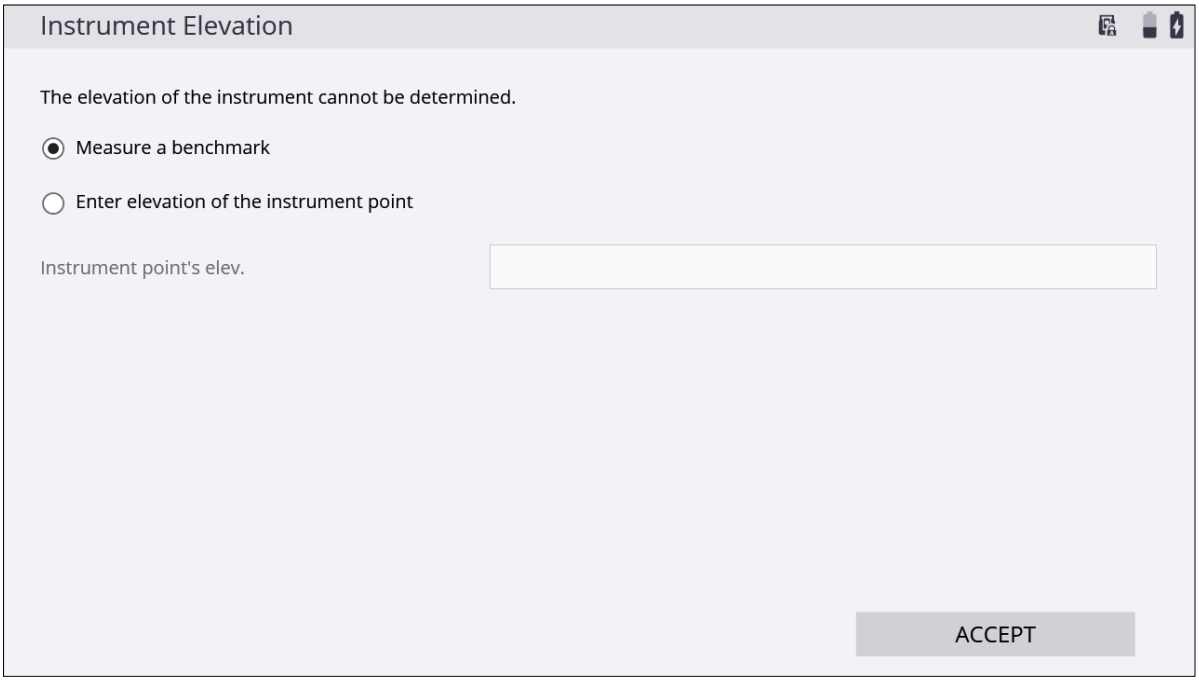
Using the last station setup

If the total station has not moved since being setup last, you can connect to the instrument and select Use last setup. This uses the same station setup parameters from the last time the instrument was setup. This method is useful for performing setups after the total station has been powered down or the battery replaced, but the instrument has not moved off the tripod.



Completing a station establishment setup where the height of the setup point has not been determined

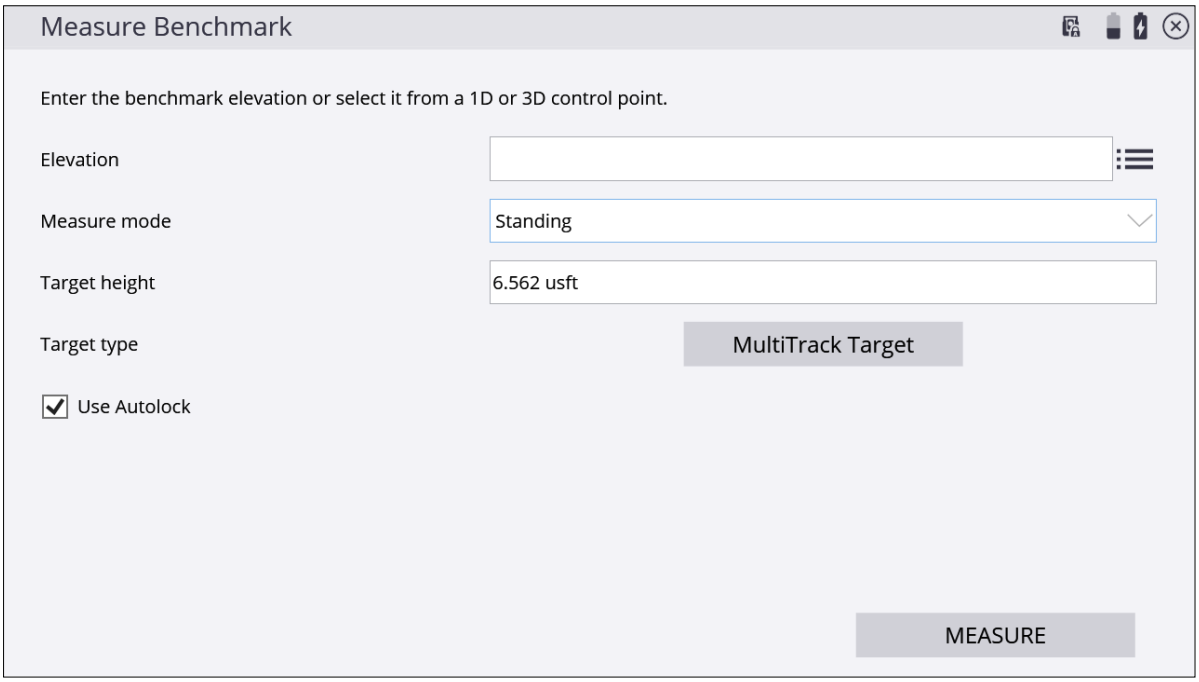
If the height of the total station has not been established at the end of the station establishment process, the software prompts you to either enter a known height for the setup point or to a height from a known benchmark location defined as a 1D or 3D point.



This occurs when:

- setting up at an arbitrary location and the reference backsight control points measured were all 2D control points.
- setting up on a known point and the known point and reference backsight control points selected were all defined as 2D control points.

The benchmark point is measured in the same way as any other control point. It is used in combination with the target height and instrument height to determine the setup point elevation.



Measuring a new control point or remeasuring a control point with a total station

Total stations require more control points around the project because of their line-of-sight dependence; therefore it may be necessary to set and measure additional control points throughout the site.

NOTE – The measure mode (Standing, Averaging, DR, DR Target, DR Averaging) selected prior to entering the Measure Control Point command is the mode that will be used to measure the control point. Ensure the proper mode is selected prior to entering the command. For the highest accuracies, it is recommended to use ***Averaging*** *mode when using a prism on a bipod, or a backsight prism on a tripod. If the control point is to be measured refelectorlessly,* ***DR Averaging*** *is the recommended mode.*

1. From the **Home** menu, tap **Measure** / **Measure Control Point**.
2. Select **New Control Points** from the menu to measure a new control point, or **Existing Control Points** to remeasure existing control points to check their position.

3. Create the control point location using a stake, a hub, or a road nail as required on the ground and then mark the stake with the name for the control point, for example, CP3.
4. Set up the Prism rod on the point and hold it steady using a bipod. The software displays the current Prism position on the map.
5. When you are ready to take the measurement, tap the **Measure** icon.

When each control point is measured, the software stores the control point data position in the control point (Control.field.csv) file for the site, and also records the measurement data into the record and report files for the work order.

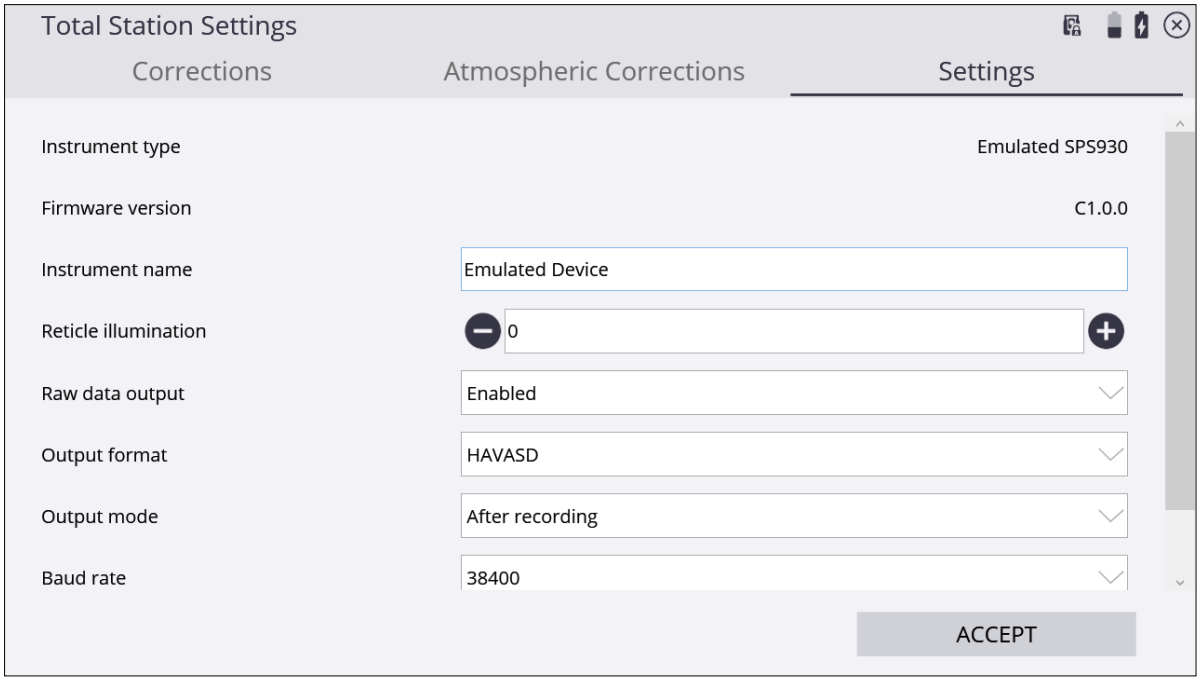
Outputting data through the COM port

In some applications, it is useful to have the controller deliver the raw measurement data or the computed coordinate data to the serial port of the controller, so that other software, for example hydrographic positioning applications, can pick it up and use it.

To set up the COM port:

1. Connect to the SPS total station and complete a station establishment.
2. From the **Home** menu, tap **Settings** and then tap **Total Station Settings**. The **Total Stations Settings** dialog appears.

3. Tap the **Settings** tab:



4. Enable the **Raw data output** option.
5. In the **Output format** list, select either HAVASD raw data or XYZ coordinates.

RAW data format	Coordinate data format
0	0
7 = HA	37 = Northing
8 = VA	38 = Easting
0 = SD	39 = Elevation

The first line of data is always a 0, which is a status tag indicating a complete measurement record.

Each line of data is terminated with a carriage return and line feed.

NOTE – Coordinate Data output is always in meters, regardless of the units selected in the Site.

6. In the **Output mode** list, select either the After recording (every time you tap **Record**) option or the Continuous (every time the total station has a new measurement) option.
7. In the **COM Port** list, select the controller port through which to send the data.
8. In the **Baud rate** list, select the rate in bits per second that you want to send the data to

the COM port.

NOTE – Parity is set to 0 (No Parity), data bits is set to 8, and stop bits is set to 1. You cannot change these settings.

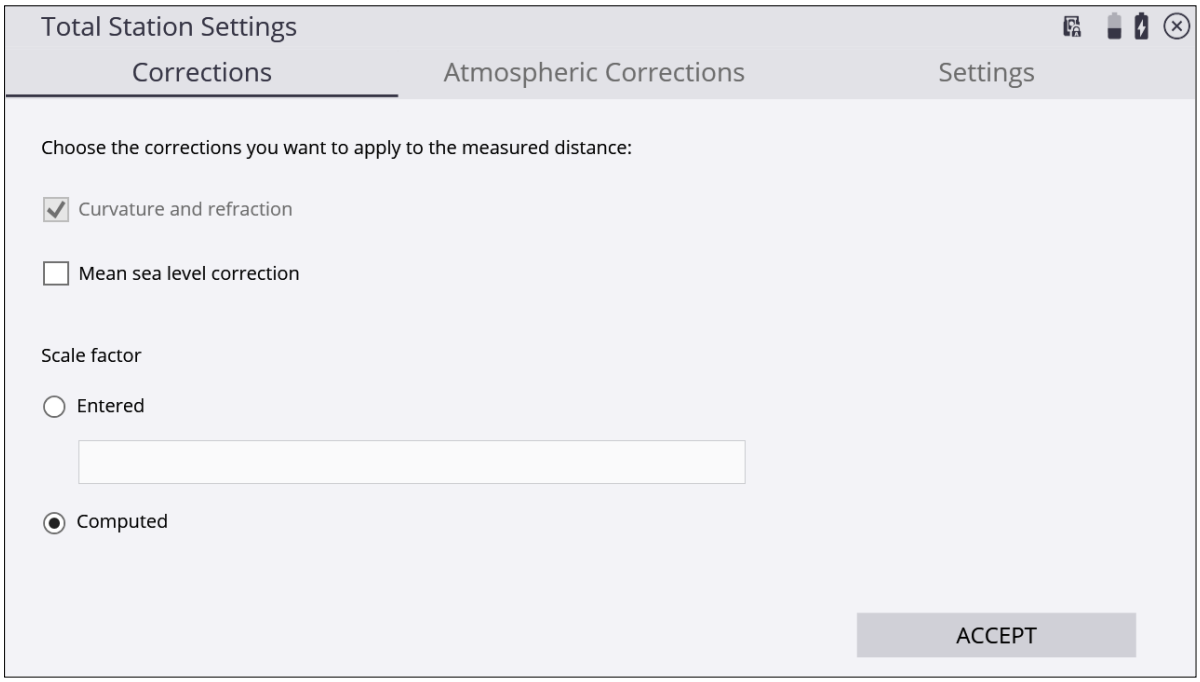
9. Tap **OK**. Once the connection is established, and measurement commences, the data will be sent to the selected COM port in the format you selected.

Computing the total station scale factor

The software can calculate a projection scale factor using a preselected coordinate system or site calibration in the site folder, for example, to measure accurately in a zone of the UTM coordinate system.

To enable the automatic project scale factor:

1. From the **Home** menu, tap **Settings**.
2. Tap **Total Station Settings**. The **Total Stations Settings** dialog appears.
3. Tap the **Corrections** tab:



4. You can enter a fixed scale factor, leave the scale factor at 1, or let the software calculate a scale factor based on the site calibration measured with GNSS or imported from Business Center. The software uses the instrument coordinates resulting out of the station establishment to calculate the correct scale factor for this instrument location in the selected coordinate system zone.

   Enabling the **Mean sea level correction** check box will account for the elevation of the instrument above mean sea level and incorporate an appropriate scale factor. This check box should be selected after at least one GNSS point is measured or a site calibration has been performed, as the GNSS elevation data from the site calibration is used to set the project elevation in the mean sea level correction calculation.

   Once the option is set to **Computed** and site measurements have been taken, the **Computed** option is locked to guarantee the consistency of the measurements.

The automatic scale factor is also written to the instrument scratch pad to apply it for machine control operations with the GCS900, Earthworks, or AccuGrade grade control systems.

12

Machine Control

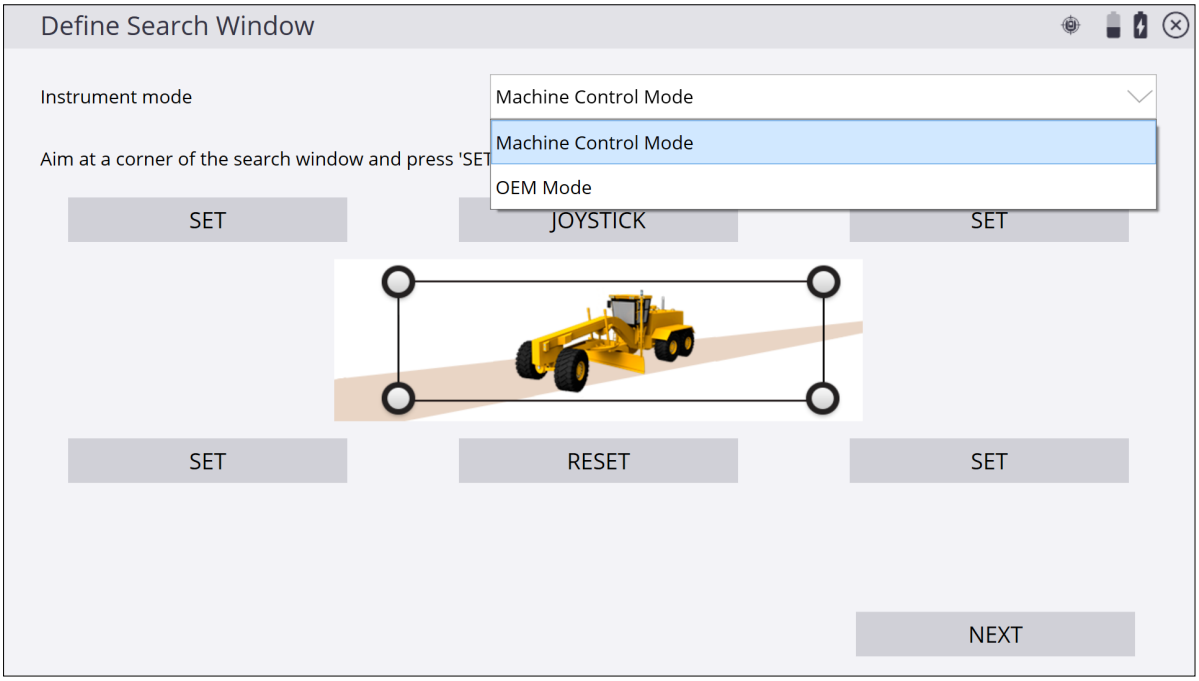
- Setting up for machine control

Setting up for machine control

From the **Home** menu, tap **Project Setup** / **Machine Control Setup**

Machine Control Setup

If the instrument has the OEM option installed, an OEM Mode is available. To set up the instrument for a Trimble GCS900 or Earthworks machine control system, select Machine Control Mode.



Defining the window for Full Search mode

When defining the window for the Full Search mode:

- Accommodate the fact that the prism can be mounted on an electric mast, and that the vertical window of operation at close range will be higher or lower than at longer ranges.
- Ensure that you set the window of operation to cover the extents of all grading operations to be completed from that location's instrument setup.
- Ensure that if the machine is taken outside that window, to turn around or to bench on a known height point, those locations are also included in the search window.

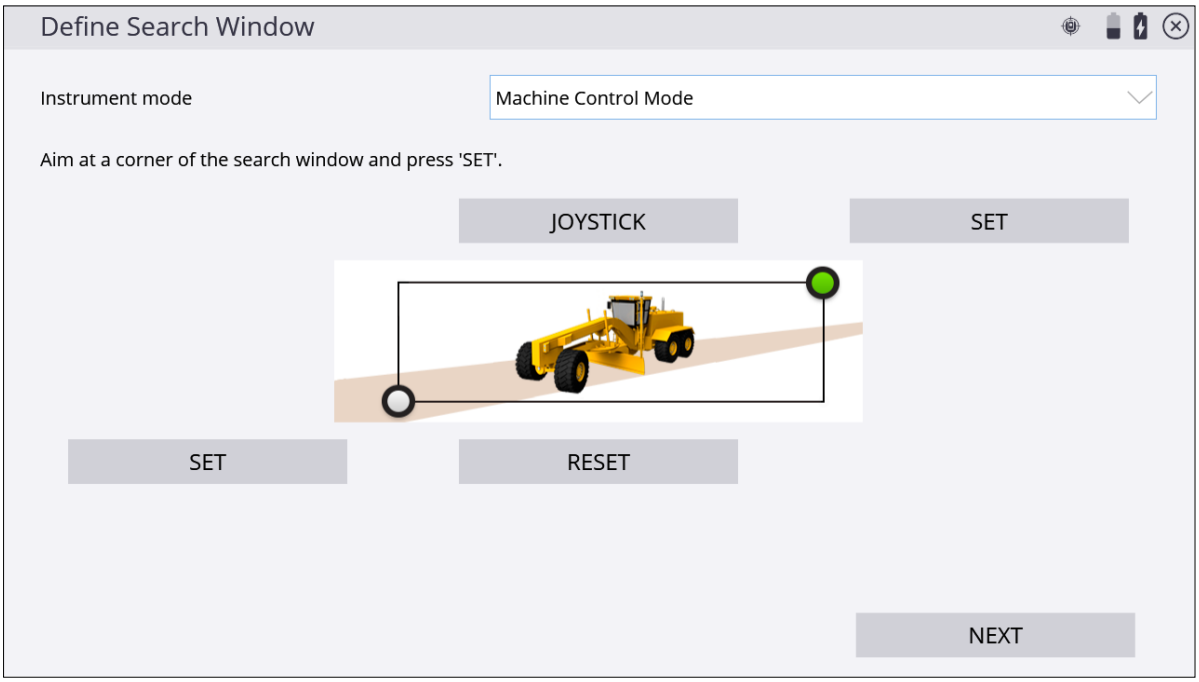
You can define either the upper left/lower right extents of your search window or the upper right/lower left extents of your search window. Ensure that you take a good look at

any high or low points in the area where you will be working and then define your search window to what most closely resembles the actual working area in front of the instrument.

- Point the total station to an upper left location and then to a lower right location; tap the appropriate **Set** button at each pointing.
- Point the total station to an upper right location and then to a lower left location, tapping the appropriate **Set** button at each pointing.

You can use the joystick function, or physically turn the total station by hand to aim at the corner points. To use the joystick to set the corner points, tap **JOYSTICK** and use the joystick commands to point the total station to one corner. When the total station is pointed in the correct location, tap **ACCEPT** in the **Joystick** window to return to the **Define Search Window**, and then hit the appropriate corner's **Set** button. Repeat this process for the opposite corner.

As soon as the first corner is defined, the only option available in the software is to define the opposite corner. For example, if you define the upper right extent first, the lower left corner only is made available to define. If the lower right corner is set first, then the upper left corner only is made available to define.



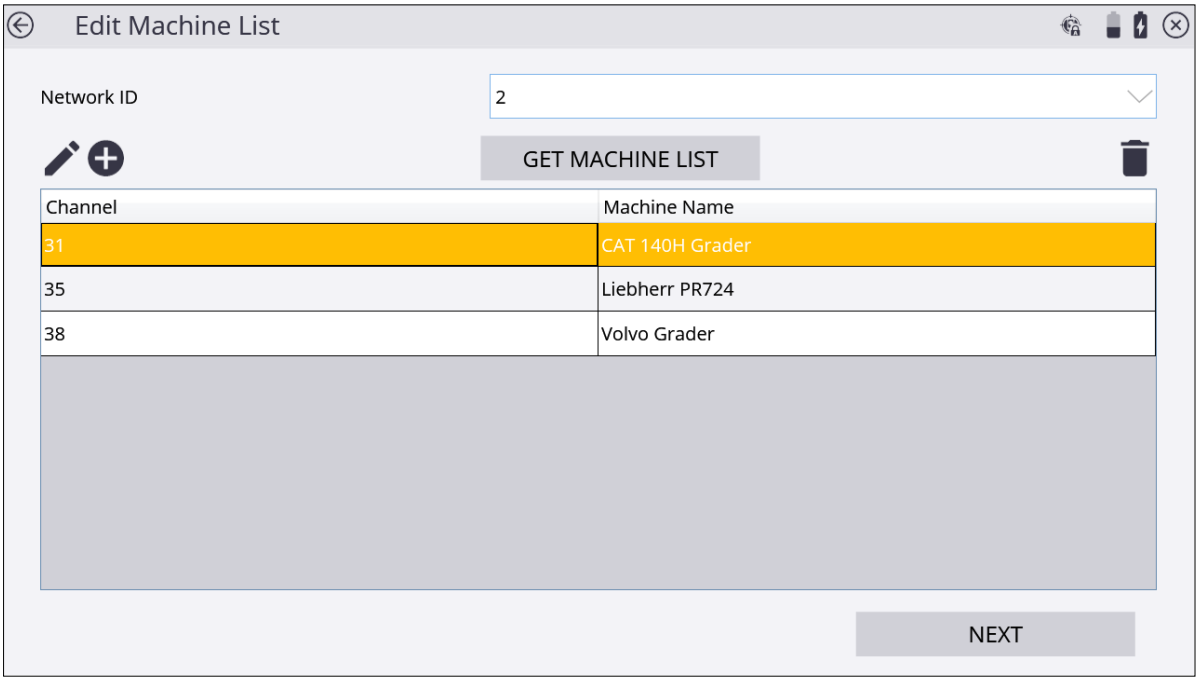
A single SPSx30 total station can be shared by multiple machines and a single SPS robotic rover pole. Only one user can use the total station at a time, but when not in use, it continually scans a predefined list of radio channels assigned to different machines and the last used site positioning system channel in sequence, looking to see if anyone needs access to the total station. When a machine or site positioning system calls the total station, it connects and provides a positioning service. When completed, the total station

returns to Standby mode and becomes available to the radio channels previously loaded to the instrument. Note that this process is made easier in Siteworks software by tapping the Disconnect button in the Project Setup menu after the Siteworks uses are complete. To facilitate this process, in the set up for machine control process, you can create a list of machines, each with a name and a specified radio channel, for example:

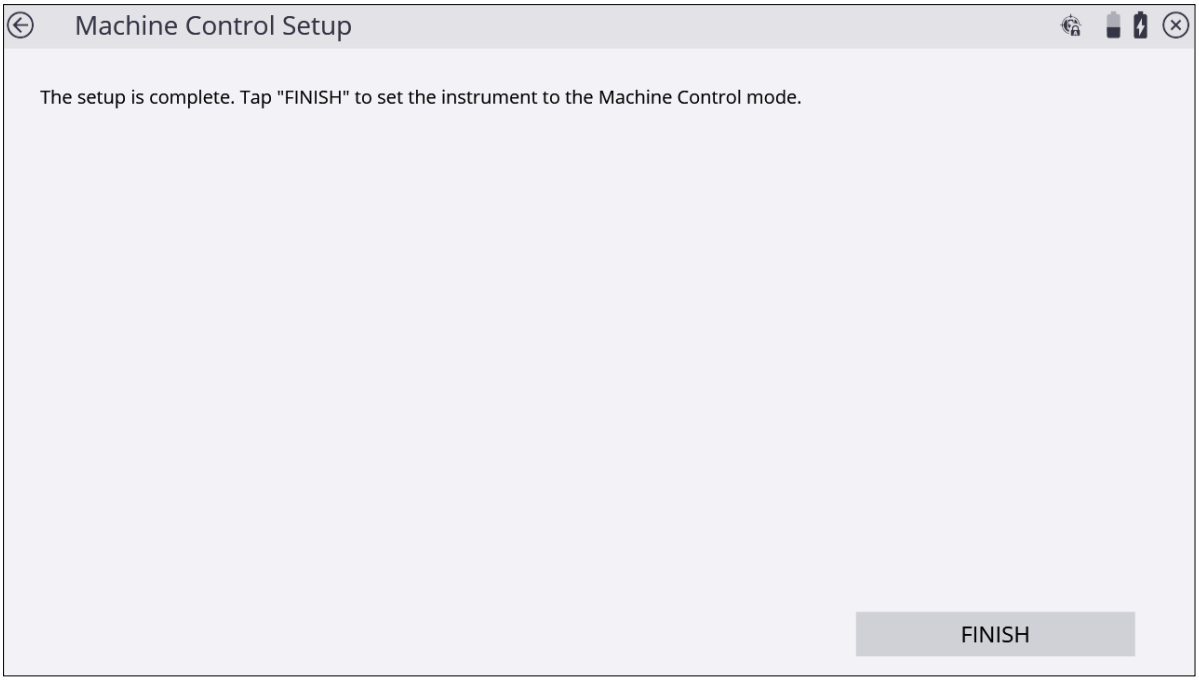
- CAT 140H Grader 1 33
- CAT D6 Dozer 1 34

The machine list is scanned sequentially whenever the total station is placed in Standby mode for machine control. The more entries in the list, the longer the scan takes to complete a full cycle and therefore the slower the initial connection will be with the total station. If you are not continually using the entries in the list in earthmoving operations, delete the entries from the list to speed up initial connection times.

Tap the + icon to create a machine in the machine list, enter a machine name and assign a network channel. Also, select a Network ID for the instrument.



This completes the instrument setup. The total station goes into Standby mode waiting for a machine to connect to it. Tap NEXT to confirm the machine control setup:



If you are tracking a machine and the target becomes lost for any reason, the total station continues to turn at a constant horizontal and vertical speed for a short period, in the expectation that the target will appear on the same trajectory when it clears the obstruction. A line-of-sight obstruction, a passing vehicle, or simply losing the target because of movement speed can cause the total station to lose the target. When the target is 'lost', that is, the above method failed to reacquire the target, the total station searches for the target. The total station has two search modes:

Mode	Description
Quick Search	Initiated as soon as the target is 'lost'. This search is centered on the last known location, and inside a window defined by a horizontal angle width (for example, 15°) and a vertical angle width (for example, 15°). The machine control software sets the values for the search window.
Full Search	This search looks inside a larger window, defined by the dialog shown previously. When the Quick Search mode fails to locate the target, the Full Search mode is initiated.

If you are operating at the total station, point it at the MT900 target on the machine. When the machine connects to the total station, this speeds up the initial search and lock-on process for the total station as it instantly “sees” the target and locks to it. The total station goes into Standby mode for machine control and then it scans the machine list in chronological order until a machine is found calling the instrument on one of the channels.

The software now disconnects from the total station. Move to the machine and initiate connection to the total station.

13

Advanced Total Station Features

Direct reflex (DR) technology is included in all SPSx30 Universal Total Stations as well as all SPSx20 Construction Total Stations. This technology offers many advantages such as stockpile scanning for accurate volume calculations.

The range varies depending on the total station you are using:

- The SPS620 and SPS720 total stations offer DR Standard technology, which enables you to measure reflectorlessly, to a maximum distance of 150 m (492 feet).
- SPSx30 total stations offer DR+ technology, allowing reflectorlessly measurement of objects up to 1,600 meters (5,249 feet) from the instrument.

Scanning stockpiles

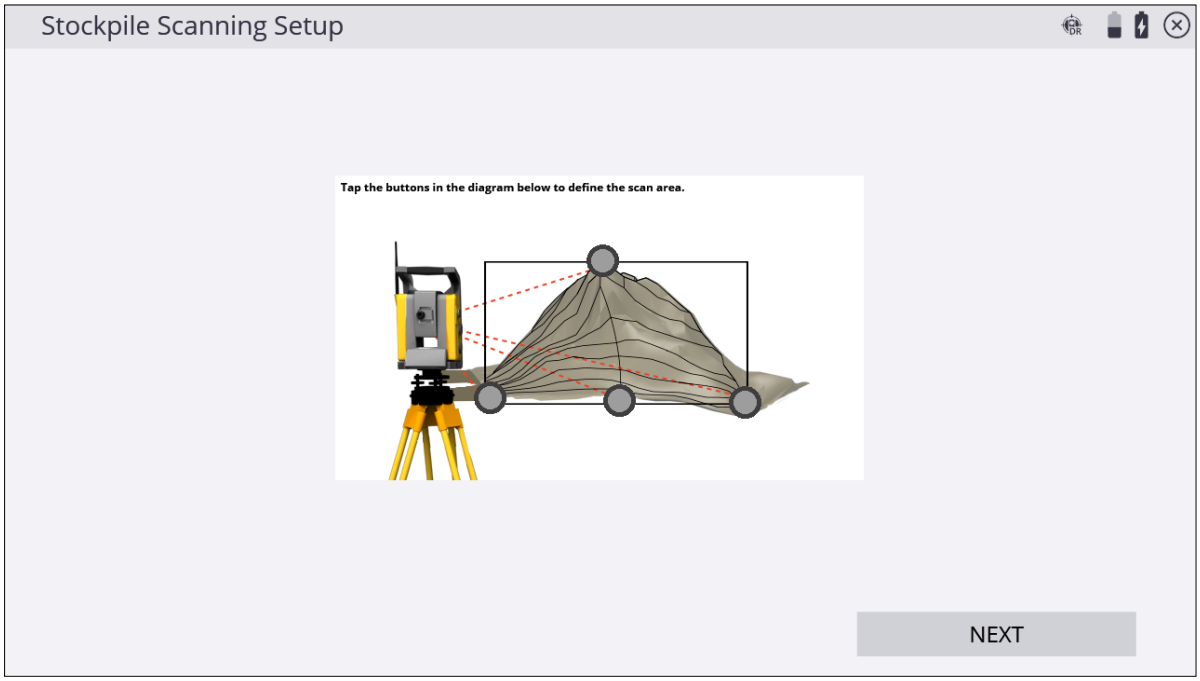
The Stockpile scan is designed to collect very accurate data for reporting on the volume of material that you have or that has been excavated. By scanning the stockpile or excavation, the need to place a worker in potentially unsafe conditions is eliminated. Reflectorless measurement technology enables you to set up the total station and measure to surfaces without using a target or prism.

Given the line-of-sight restrictions when using a total station, you need to perform a minimum of two station setups to collect all sides of the stockpile:

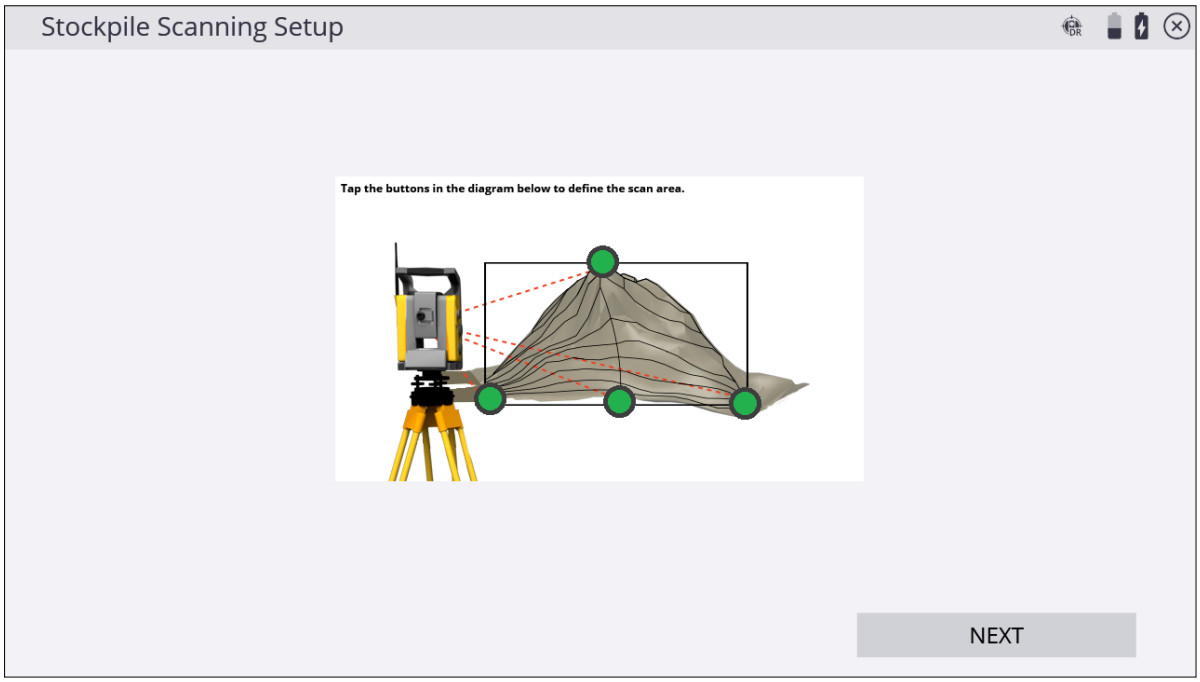
1. From your first setup, measure new control points around the stockpile in locations that will give you the fewest setups and the best vantage points to scan the largest surface area of the stockpile. This enables you to collect all sides of the stockpile and have the surface points correctly related to each other.
2. Once the whole stockpile has been scanned, define a volume boundary in the **Cogo** menu by tapping the points to be used to define the boundary, and then perform a volume calculation in the field.

To set up stockpile scanning:

1. From the **Home** menu, tap **Measure** and then tap **Stockpile Scanning**.
2. The software sets the instrument to DR mode, and then the following screen appears:

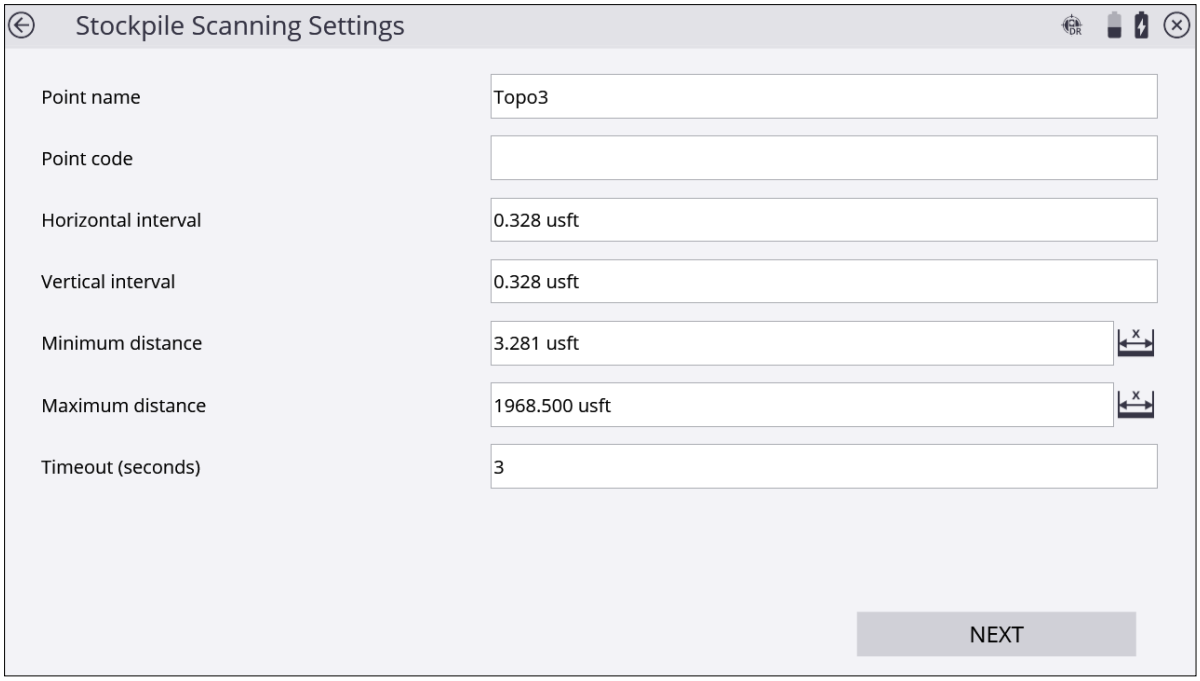


3. Define the shape of your stockpile. Start by tapping the gray dot at the highest point of the stockpile image on the screen.
4. Manually sight your instrument to the highest point of the stockpile and then tap **Measure**. The **Point Mode** dialog appears.
5. If required, enter a point code and point name and then tap **OK**. The **Stockpile Scanning Setup** dialog reappears.
6. Repeat Step 4 through Step 6 to define the bottom left, bottom right, and bottom most points of your stockpile. Once successfully measured, the points appear green:

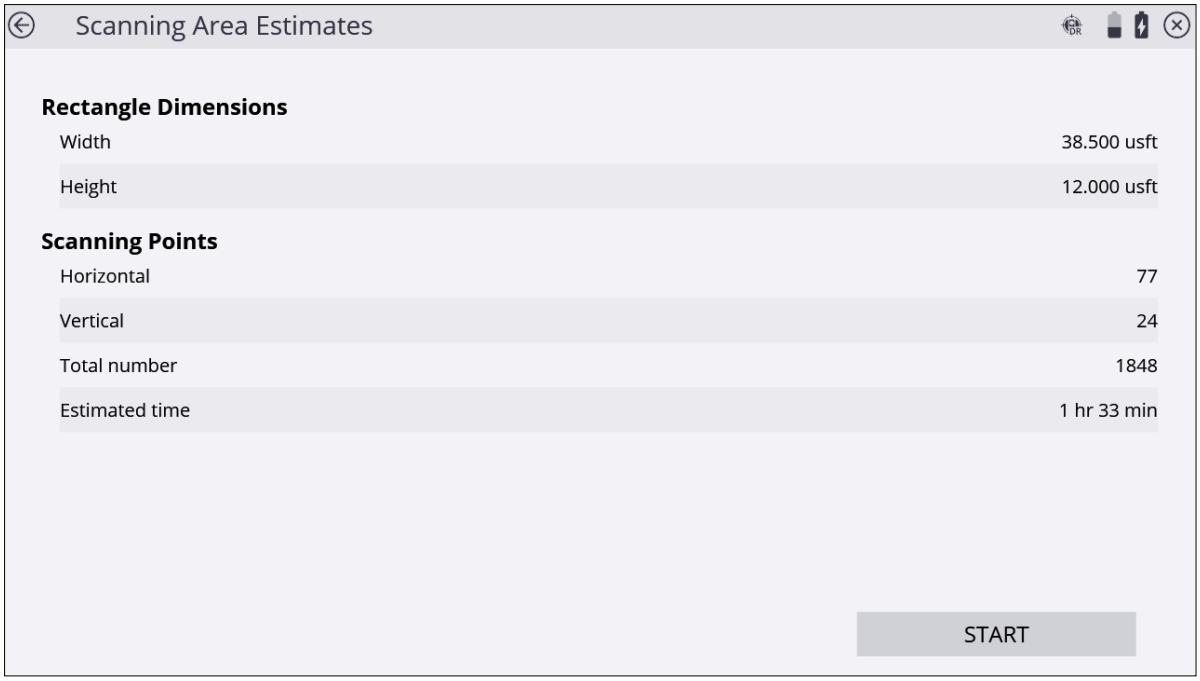


NOTE – This feature and the way that the corner points are configured in Siteworks software is based upon setting the target points on a vertical plane. The lower left and right points set the vertical edges of the plane, and the top and center bottom points define the upper and lower horizontal edges of the plane. If this feature is used at long range, along a gently sloping surface, and/or with a large vertical distance between top and bottom points, it will result in a non-uniform "fan-shaped" spacing of points, as the scan continues to measure points located upwards along the plane, and further from the instrument. This feature is not a substitute for traditional 3D laser scanning methods.

7. Tap **Next**. The **Stockpile Scanning Settings** dialog appears:



8. Enter a point name and code and set the horizontal and vertical distance intervals. Setting the distance intervals low results in more measured points; setting the distance intervals high results in fewer measured points. Also, having a smaller interval (more points) will result in a longer scan. When entering these values, keep the size and shape of the stockpile in mind. Define your minimum and maximum distances for point data collection and then tap **Next**. Correct use of these settings helps you collect only the relevant points in the field on your stockpile. If a point falls outside of these distance values, it will not be measured. Consider setting the minimum distance to be approximately the distance from the instrument to the bottom of the stock pile (closest desired point to be measured), and the maximum distance to be near the top of the stockpile (furthest desired point to be measured). The software will default to the recommended Timeout value depending on which instrument is being used.
9. The **Scanning Area Estimates** dialog displays the total number of points to be collected as a result of the dimension and distance intervals previously entered and an estimation of the time it will take to record the points. This time is an estimate only and the reflectivity of the material, distances involved, and the type of instrument used alters the total time once the scan has started.



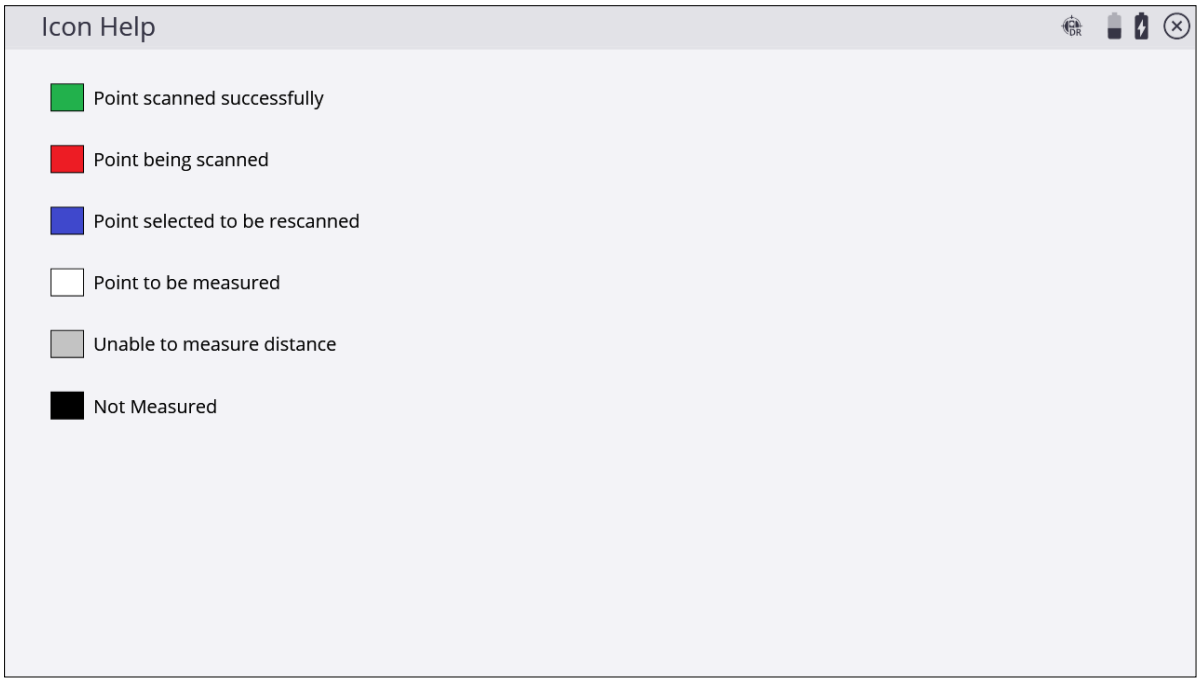
CAUTION – The values stored in Step 8 are overwritten by the minimum and maximum values set in the DR Target Settings dialog. Ensure that you check the values set in the DR Target Settings dialog to eliminate confusion by another operator when they are setting the Stockpile Scan settings. Setting the minimum distance to 2 meters (6.56 feet) and maximum distance to 1600 meters (5,249 feet) in the DR Target Settings dialog will eliminate any errors when setting the Stockpile Scan settings.

TIP – When working at the maximum ranges of the DR technology, increase the Timeout Setting. When adjusting the timeout settings, be aware the material being measured affects the strength of the response signal. Also, the instrument's technology affects the time it takes to record each individual point.

10. Either tap the back arrow to go back and change any settings before starting the scan or tap Start to start the scan.
11. A grid is displayed as the scan proceeds. Scan time remaining is displayed in the Estimated Time Remaining field. If you need to change the minimum and maximum distance settings, tap Pause and then tap  . Change the minimum and maximum settings and then tap Next. The scan will resume.
12. Tap  in the map control bar on the right to display the map view so you can see the

points being collected.

TIP – For an explanation of the different grid sector colors, tap in the upper right of the screen. The following screen appears:



13. To rescan any cells, tap **OK** to return to the measured cells display. Tap the associated grid sectors with your stylus. Selected sectors turn blue. Once the selection is made, tap **Rescan**. Repeat the rescan function until results are satisfactory and then tap **Finish**.

Adjusting the total station

All total stations require regular and routine checks and adjustments to deliver optimum results. All Trimble total stations allow fully accurate measurements to be made with a single pointing to a target. To achieve those results, the total station stores its current adjustment values internally, and then corrects all measured data accordingly. For accurate measurements to be made, the current adjustment values need to be determined and stored in the memory. Total station adjustments are required because of the optical-mechanical design of the instrument. The following conditions can move the optics and mechanics out of adjustment:

- Shipping and handling
- Bumps and knocks
- Temperature and pressure changes
- Storage conditions
- General wear and tear of mechanics

To start the calibration:

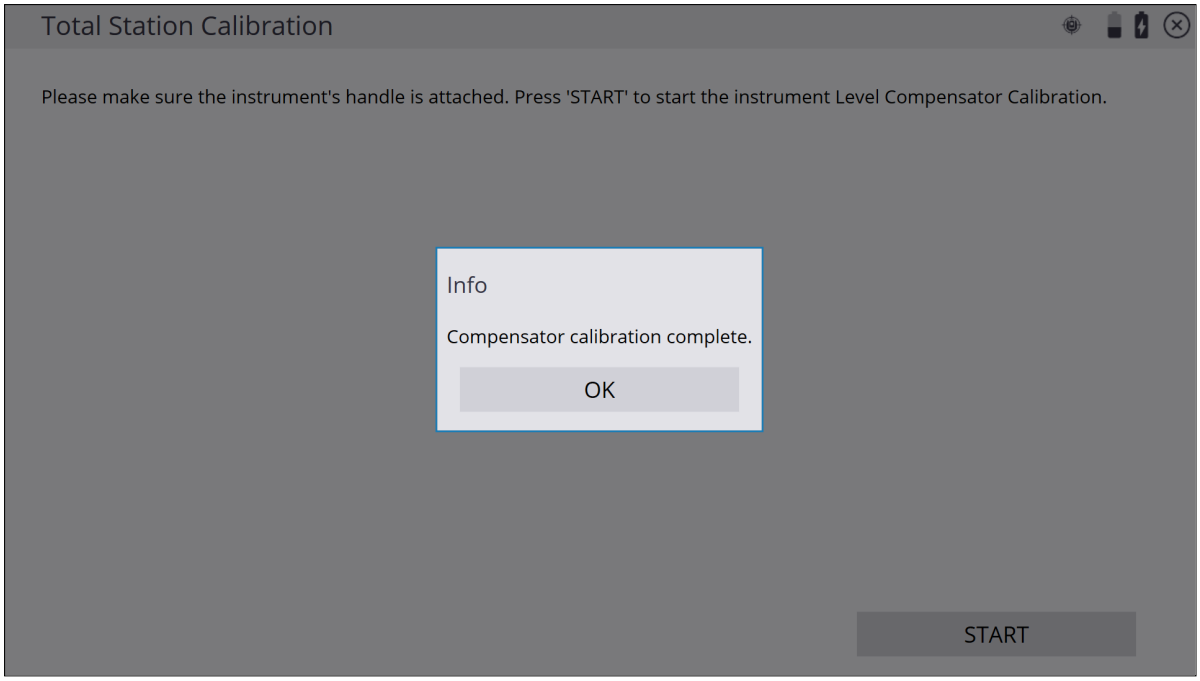
1. From the **Home** menu, tap **Settings** / **Total Station Calibration**.

   ### Compensator Calibration

   ⚠ CAUTION – You must perform this calibration before the HA VA Collimation and Tracker Collimation. The compensator does not need to be calibrated every time the other collimations are performed, but if the Compensator Calibration is performed, you should immediately perform the HA VA Collimation and Tracker Collimation. Performing the Compensator Calibration negates the validity of the values of the errors found from previous HA VA Collimations and Tracker Collimations.

   The SPS family of motorized total stations is all equipped with a dual-axis compensator. The compensator is active when the total station is switched on. You should periodically calibrate the compensator to adjust for any minor changes in the total station caused by normal wear and tear, as well as shipping or temperature variations. It is extremely important to perform this calibration when you are working within a very tight tolerance range. You should also perform this calibration whenever the highest accuracies are needed.

2. Tap **Start** to initiate the instrument Level Compensator Calibration:

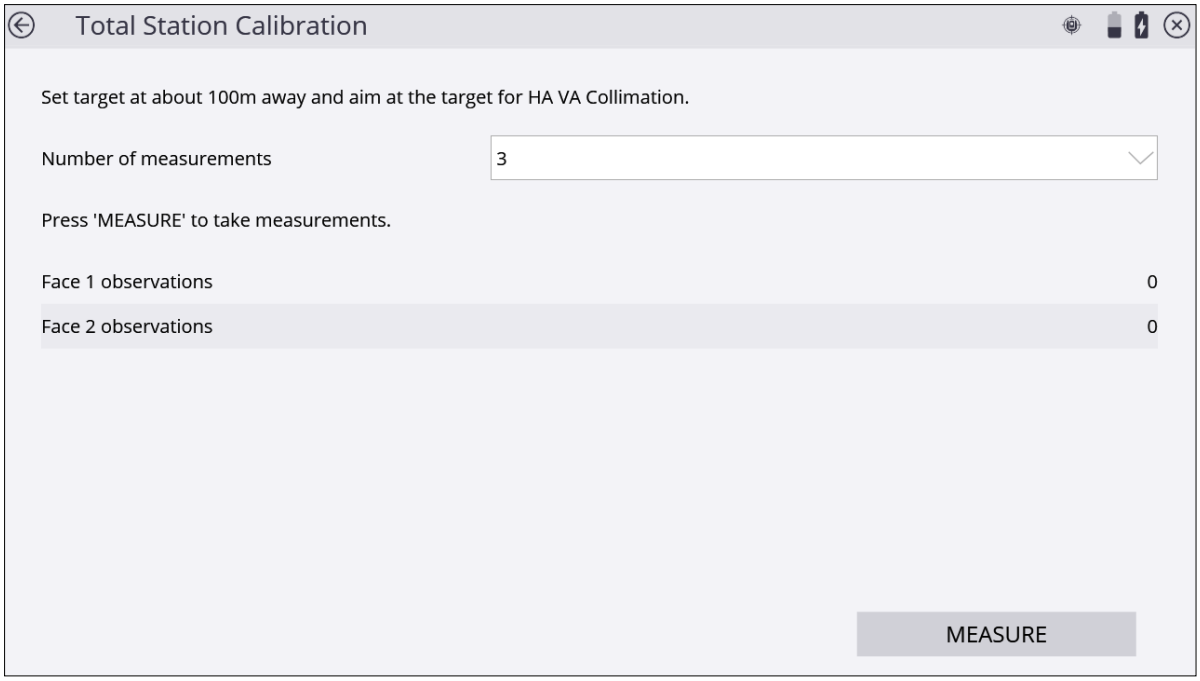


3. Tap **OK**.

HA VA Collimation test

You should perform this test to a target that can be easily bisected with both the horizontal and vertical cross-hair, placed at a location at least 100 m (328 feet) from the total station, and at approximately the same elevation as the total station telescope. The target can be any object including a road sign, window frame, or an adhesive prism target. The test involves taking a series of HA VA measurements to the target in both instrument faces, to generate a mean or averaged pointing in face 1 and face 2, from which the difference between face 1 and face 2 readings can be determined. The difference between the two face readings is known as the collimation error. In the horizontal axis, the collimation error has little effect on measurements. However in the vertical axis, if not corrected for, the collimation error will result in erroneous elevation values for all measured points.

4. Enter the number of measurements, aim at the target and then tap **Measure**:



The collimation test computes the collimation error, stores the error inside the total station and then corrects all subsequent measurements for that error before displaying them on the screen or storing them in memory.

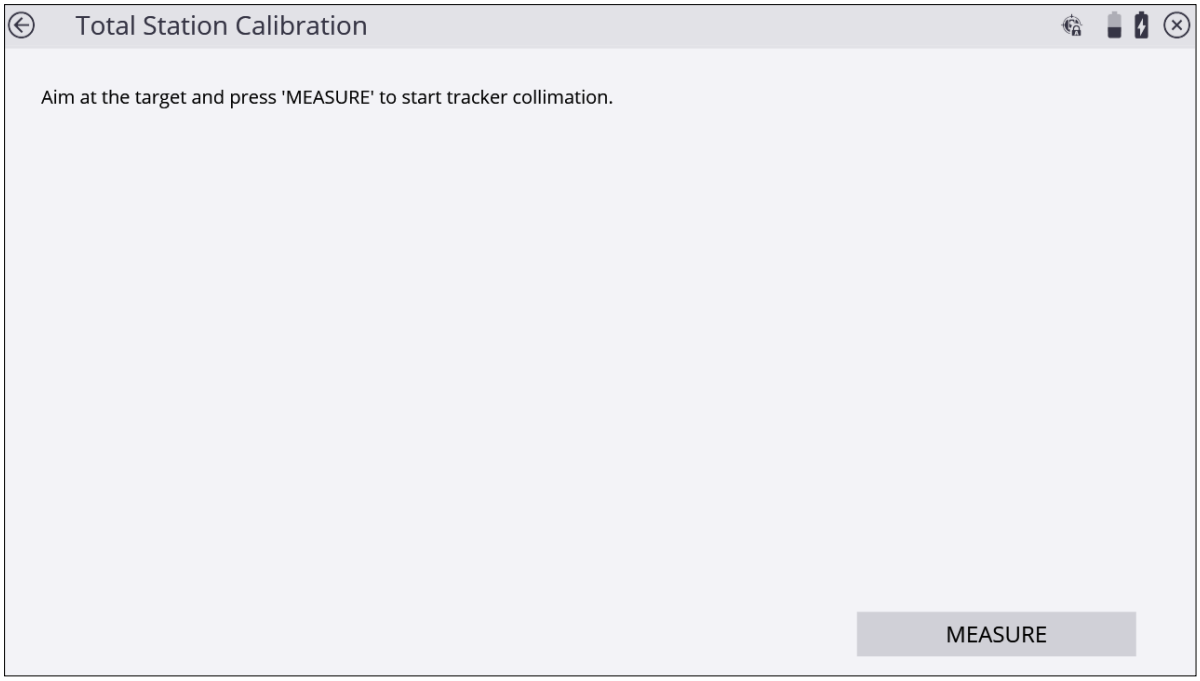
5. Tap **Next**.

Tracker Collimation test

You should perform this test towards a prism or active target set up at a distance of around 100 m and at approximately the same elevation as the total station telescope. Ideally, perform the test at the approximate range that subsequent measurements will be made.

The test involves the total station locking onto and measuring an average position over a period of time in both faces to determine any misalignment of the tracker in relationship to the telescope cross-hair. If not corrected for, this error results in erroneous position determination in both horizontal and vertical axes, and also between measurements made with and without Autolock technology. Once measured, the error is stored in the total station, and is used to correct all subsequently measured positions.

6. Aim at the target and tap **Measure**:



NOTE – There can be two reasons for significant change between old and new values: (1) The total station has received a knock or bump in transit that may need a service correction, or (2) there has been an observation error.

If you suspect an observation error, repeat the process. If the values are repeated, you may want to contact an authorized Trimble Service Center for advice. When the values exceed a certain level, you will be advised to send the total station to an approved Trimble Service Center for recalibration.

NOTE – The values displayed when new will be close to zero, but over time these are expected to change. Non-zero values are no cause for concern; however, sudden large changes should be cause for concern because they indicate misuse, abuse, or transportation problems. For full details of the instrumentation errors, refer to your instrument manual.

7. Tap **Finish**.

Measure Control Network

The Siteworks software includes the Measure Control Network function, which enables you to configure and measure rounds of angles to different control points of a network or traverse. When you measure at least two rounds to a control point, the software will calculate the standard deviation for each foresight and backsight target (accuracy) and the standard deviation of the mean (precision). This enables you to evaluate the quality of the measurements in the field.

To use this function, you must purchase and install the Advanced Measurement module. When connected to an SPS instrument, the **Total Station Setup** menu contains a new option called **Measure Control Network**. To adjust a traverse or control point network measured with this feature, you need the Total Station Processing module for the Business Center – HCE software. The Siteworks software exports the RAW data to a DC file, which is then imported into the Business Center – HCE software.

Measuring rounds of angles

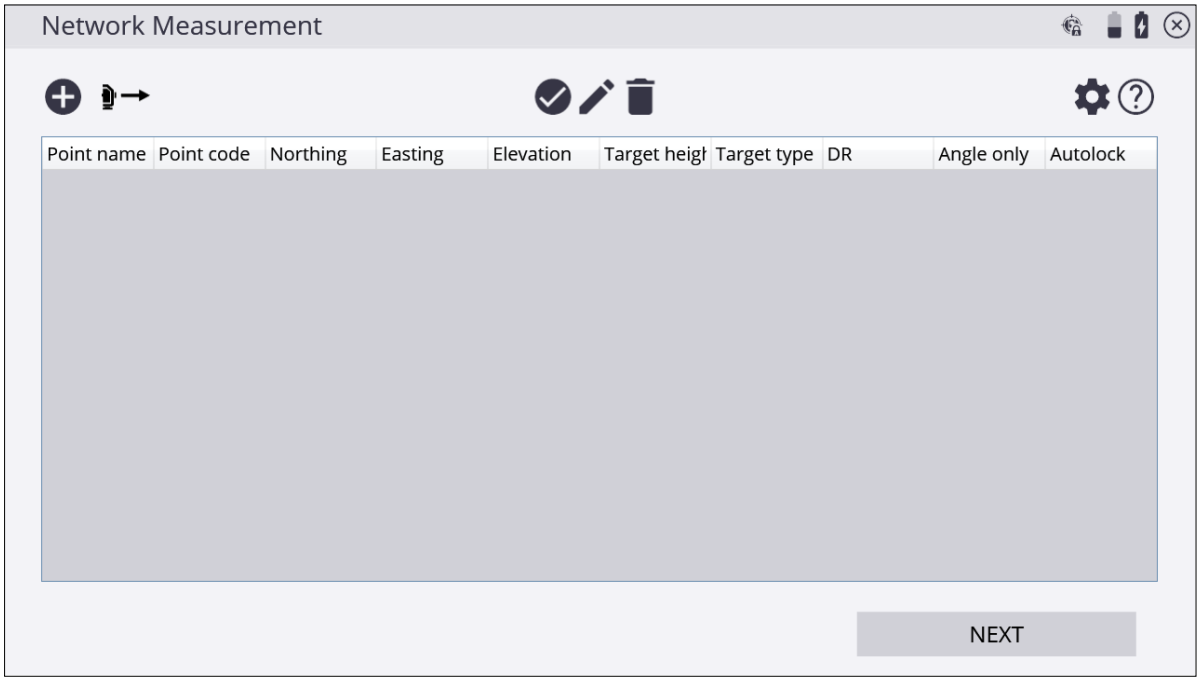
To start the network measurement:

1. Open a project and work order that contains your existing control points and connect the controller to an instrument.

   Optionally, you can perform an instrument setup before starting the network measurement. This enables you, during the configuration of the network measurement, to check the point locations to verify if the prisms were set up over the correct control point.

2. Tap **Home** / **Project Setup** / **Total Station Setup** and select **Measure Control Network**. Tap **Accept**.

3. Use the following screen to configure the measurements for a control point:

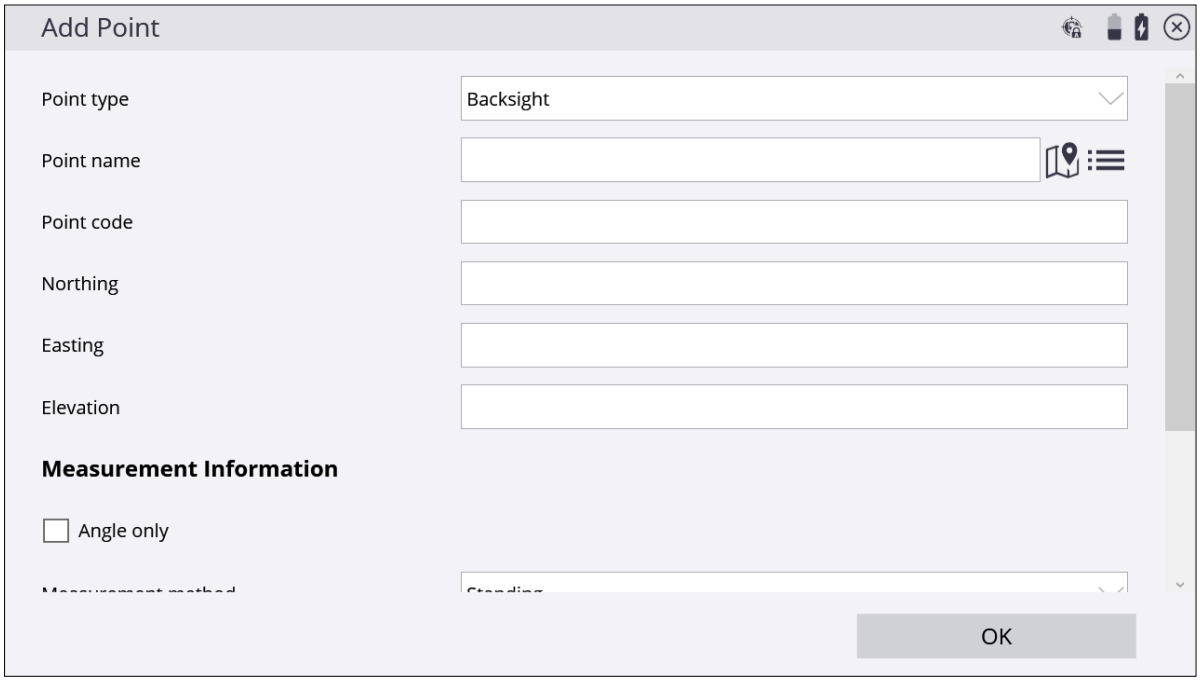


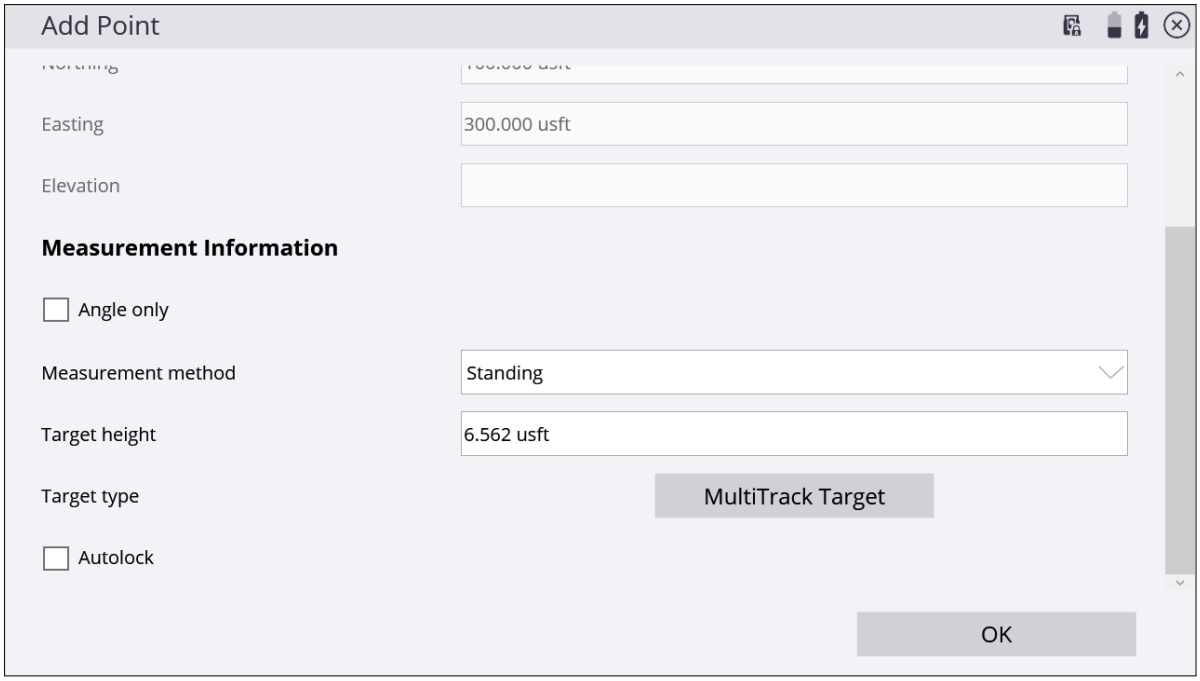
4. The **Network Measurement** dialog contains the following buttons:

Tap this button ...	to ...
	add a new point to the list
	measure and add a foresight point to the list
	check a point location.
	edit point information.
	delete a point from the list.
	edit the settings for network measurement.

5. To add an instrument point followed by all the backsight points for your setup, tap  .

   If the Control Point is not included in the Control Point file yet, then enter the coordinates in the following dialog:





6. In the **Measurement Information** group, you can decide if an individual control point should only be measured reflectorlessly or angle only, and can enter the correct target height, target type, and target ID.
7. When selecting the **Autolock** option, the instrument will automatically lock on the prism after the first set. If you leave this option cleared, you will manually need to aim the total station at the prism each time.

   TIP – When adding the next point, tap **Use Last Settings** to copy the settings from the previous point so that you do not have to re-enter all the values.

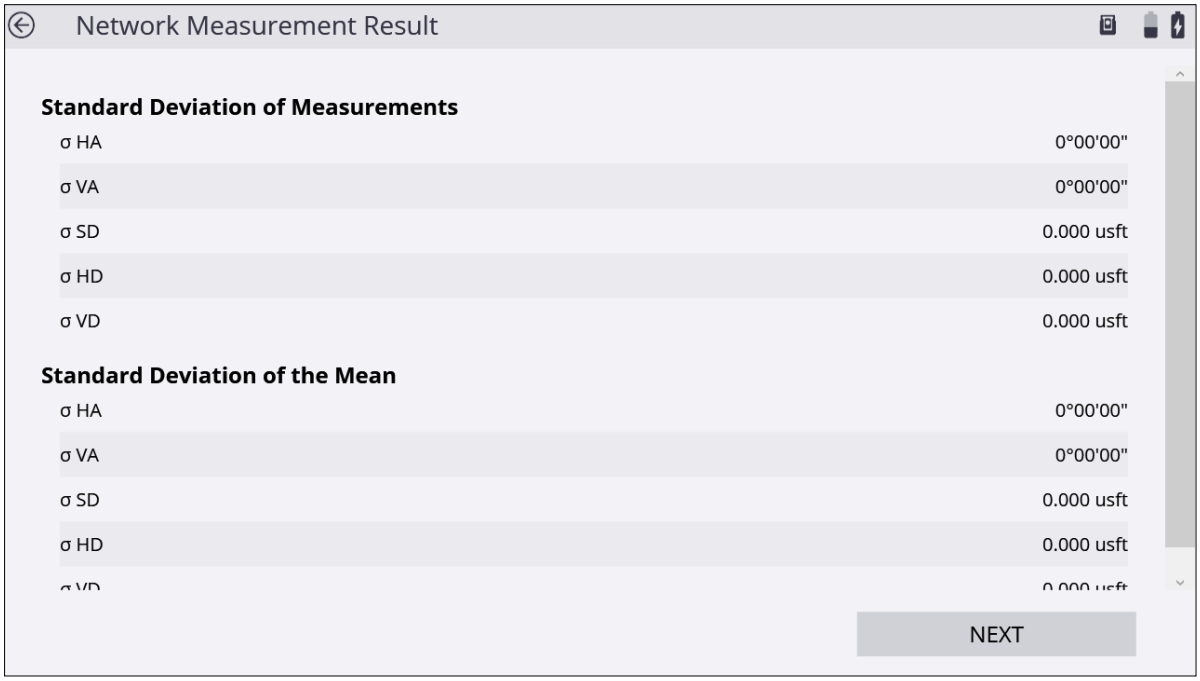
8. To add and measure new control points as foresight points, tap  . To configure the point, see Step 4 through Step 5.
9. If required, tap  to check the point location of an individual point from the list and verify that the prism is set up over the correct point.
10. Tap  to edit the configuration of each point or tap  to delete it from the list completely.
11. Once the configuration is correct, tap **Next**:

12. Tap  to configure the face and round order as follows:

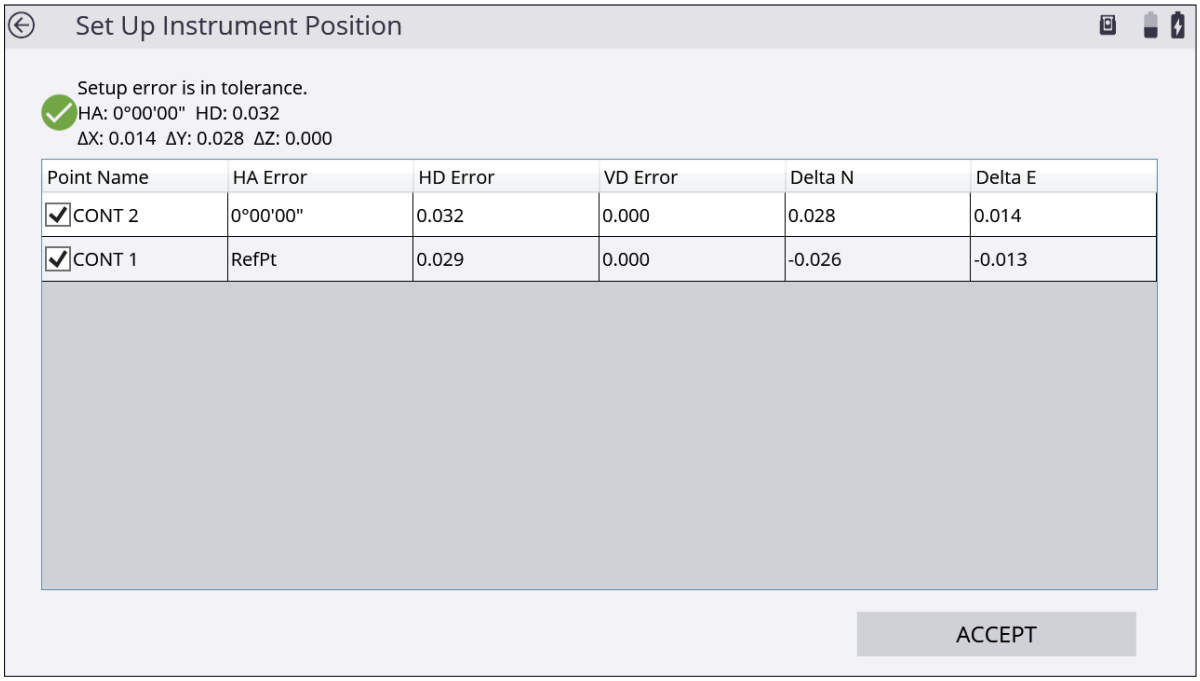
Tap ...	to set ...
F1	all targets only in face 1.
F1 F2	all targets only in face 1 first and then swaps in face 2 and measure all targets again.
F1/F2	each target in face 1 and then face 2 before going to the next target.
123...123...	all targets in face 1 from left to right then swap in face 2 and measure from left to right.
123...321...	all targets in face 1 from left to right then swap in face 2 and measure from right to left.

13. To calculate the standard deviation for each target, measure at least two rounds.
14. Tap **Next** and then tap **Next** again to access the **Measurement** screen. The Siteworks software prompts you to aim at the different targets and measure the first set. To measure the remaining sets automatically, enable Autolock.
15. While measuring the second set, the software indicates the differences between face 1 and face 2. If the measurements are within the tolerances that were entered for the differences between face 1 and face 2, the software shows the status "PASS" for that point.

16. When all of the rounds are measured, the software shows the **Standard Deviation of Measurements**, which is an indication for the accuracy of all of the measurements. The **Standard Deviation of the Mean** gives an estimate of the precision of the measurement.

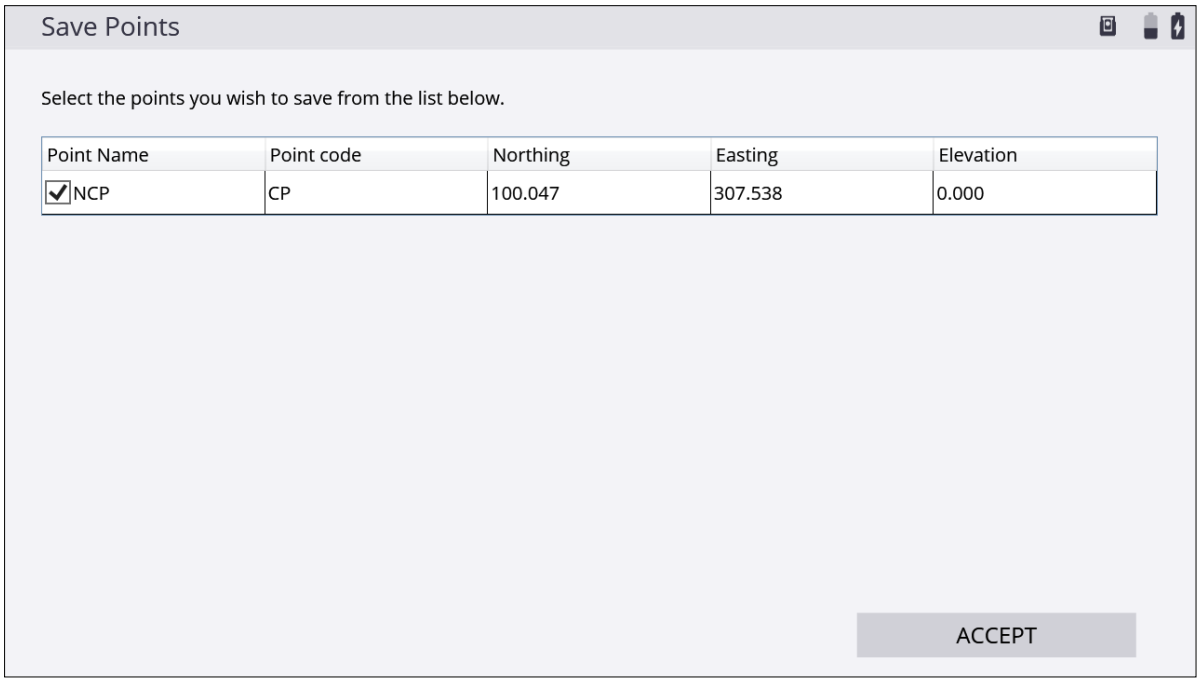


17. The next screen shows the differences between the measured rounds for each target:



18. The next screen allows you to select which control points should be stored in the

Control Point file of the site. The coordinates are a result of the average angle and distance measurements on one control point, but they do not take a traverse or network adjustment in account:



19. Once all measurements and all control points are measured, you can export the RAW data of the measurements for doing a network adjustment. Select the**Home** menu / **Data Management** / **Export Measured Data**. To adjust the network in the Business Center software, export the data with the Network Measurement (DC) option. An export for StarNet network adjustment software is also available. Use the Network measurement (XSLT). The files with the raw data are stored in the Output folder of the current work order.

14

Traverse Workflow

A *traverse* is a surveying method used by construction surveyors to establish horizontal and vertical control point networks in situations where high accuracy is needed and where GPS is unsuitable to use. The process involves placing survey stations (control points) along a path of travel, and then using the previously surveyed points as a base for observing the next point.

Traverses consist of three parts: the starting station, the intermediate stations, and the ending station. The starting and ending stations may, or may not, be the same control point. A traverse is said to be closed when it starts and ends on two sets of known control points, and open when it ends on an unknown point. Traverses can be closed with two, three, or four control points depending on traverse type, but the starting and ending stations must be set on and shot to another control point that existed prior to starting the traverse. Currently, the Siteworks software only supports closed traverses.

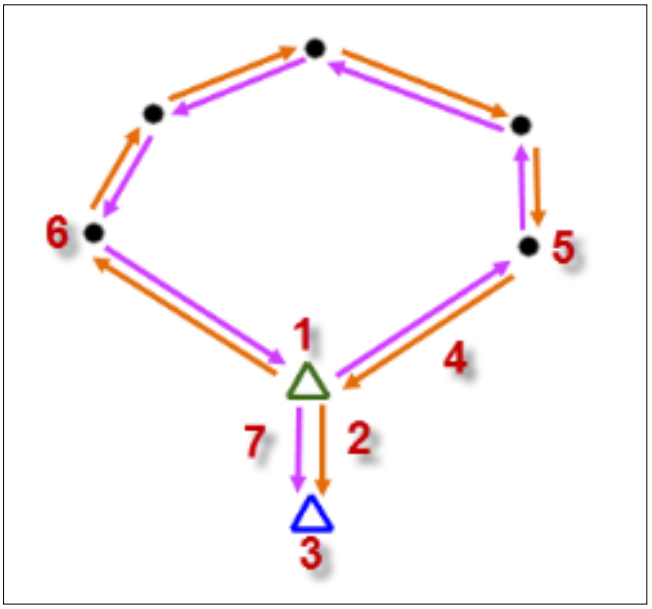
The intermediate stations are all the stations between the starting and ending station that are "set" as part of the traverse. In all configurations, the points these intermediate stations are set up on will be adjusted after the traverse measurements are completed.

By using the traverse feature in the Siteworks software, you can run any closed traverse and get final adjustment results in the field without needing to export data to other software. The adjustment results will match those calculated in a Trimble Business Center project.

Example of a closed-loop traverse

Below is an example of a typical closed-loop traverse in which the traverse starts and ends on the same control point:

- The starting station (1) is set on a control point with a backsight (orange line; 2) to another control point or known azimuth (3).
- The starting station then foresights (purple line; 4) to the first intermediate station (5).
- Each intermediate station in turn backsights (orange lines) the previous station and foresights (purple lines) the next station.
- The ending station (1) then backsights the last intermediate station (6) and foresights (7) the original control point (3) that was backsighted from the starting station.



Control points

This chapter assumes that the data collector is already connected to a total station. For a detailed description on how to connect to a total station, see Connecting to a total station, page 155.

NOTE – The traverse function only works with 3D control points.

To build a closed traverse, control points for the starting and ending stations are required. There are multiple ways to obtain control points:

- Enter and/or edit the coordinates of the point using the Enter/Edit Control Points functions.

  See The Total Station menu, page 20.

- Measure the coordinates of the point using the Measure Control Point functions.

  See Measuring a new control point with GNSS, page 142.

- Import control points from a .csv file.

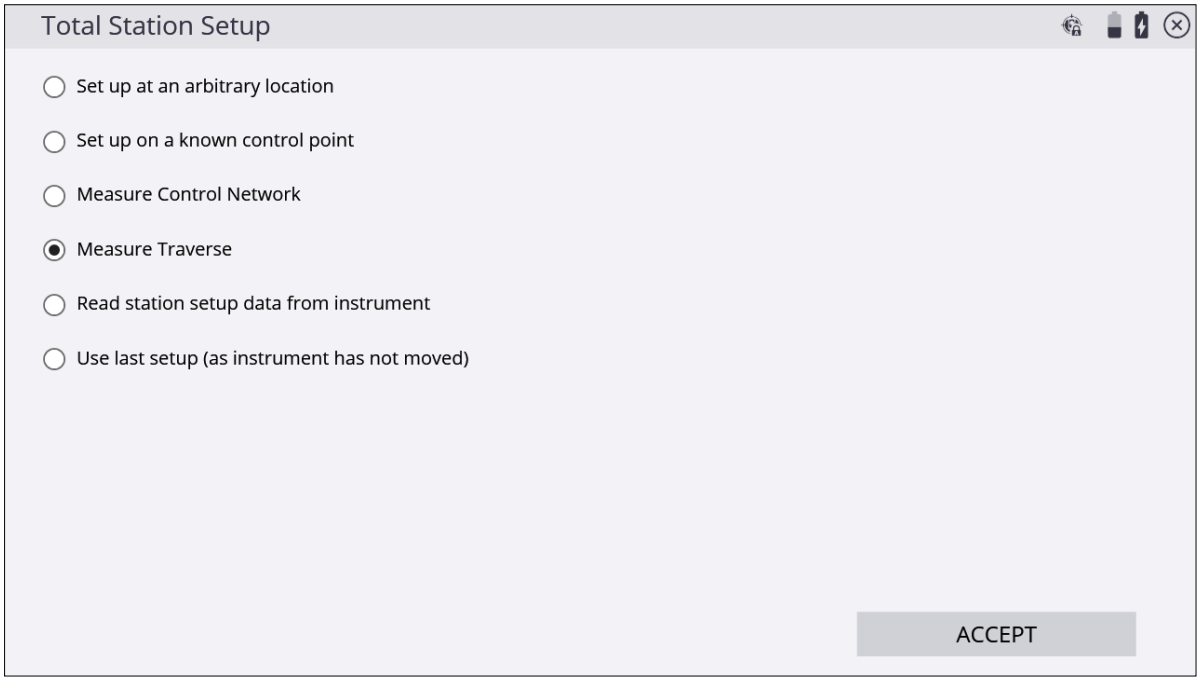
  See The Project Setup menu, page 17.

Building the traverse

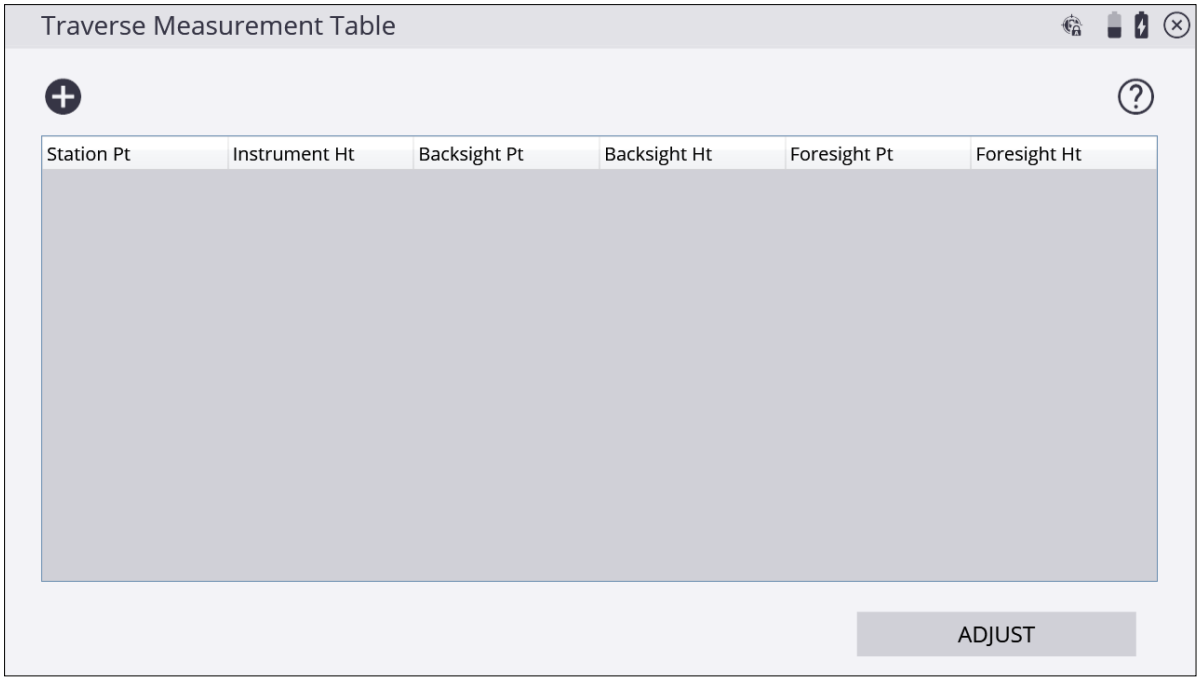
NOTE – The traverse workflow only supports closed traverses as open traverses do not allow for adjustments (distributing small parts of error) to each measured station.

To build the traverse:

1. From the **Home** menu, tap **Project Setup** and then **Total Station Setup**:



2. Select **Measure Traverse** and tap **Accept**. The following screen appears:



3. To add a traverse station, tap .

   The **Traverse Measurement Table** screen contains the following buttons:

Tap ...	to ...
	measure a new traverse station
	remeasure a traverse station
	remove the last traverse measurement
	clear traverse measurements

4. In the **Instrument Setup** window, there are two ways to define an instrument point:

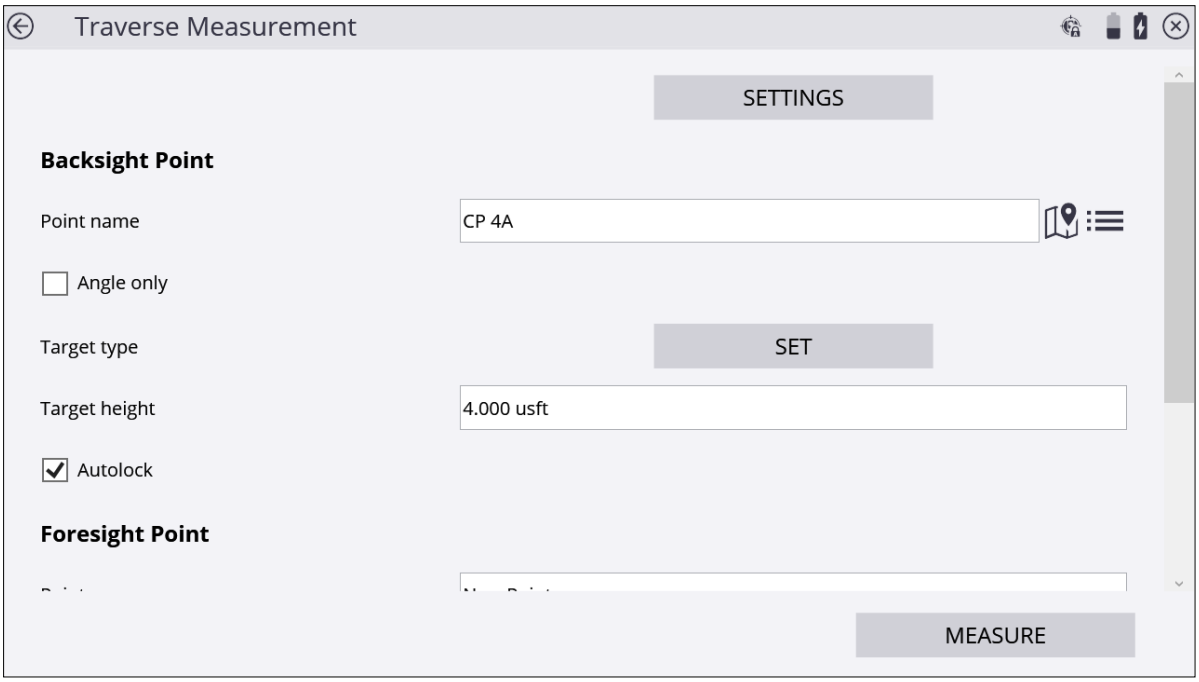
   - Select a control point/point from the map .
   - Select a control point/point from the point list .

   NOTE – You can manually enter the control point/point name.

5. After all the instrument point information is entered, tap **Next**.

6. In the **Traverse Measurement** screen, enter the backsight and foresight point information:

   a. Backsight point:



   The backsight point can be selected by two methods:

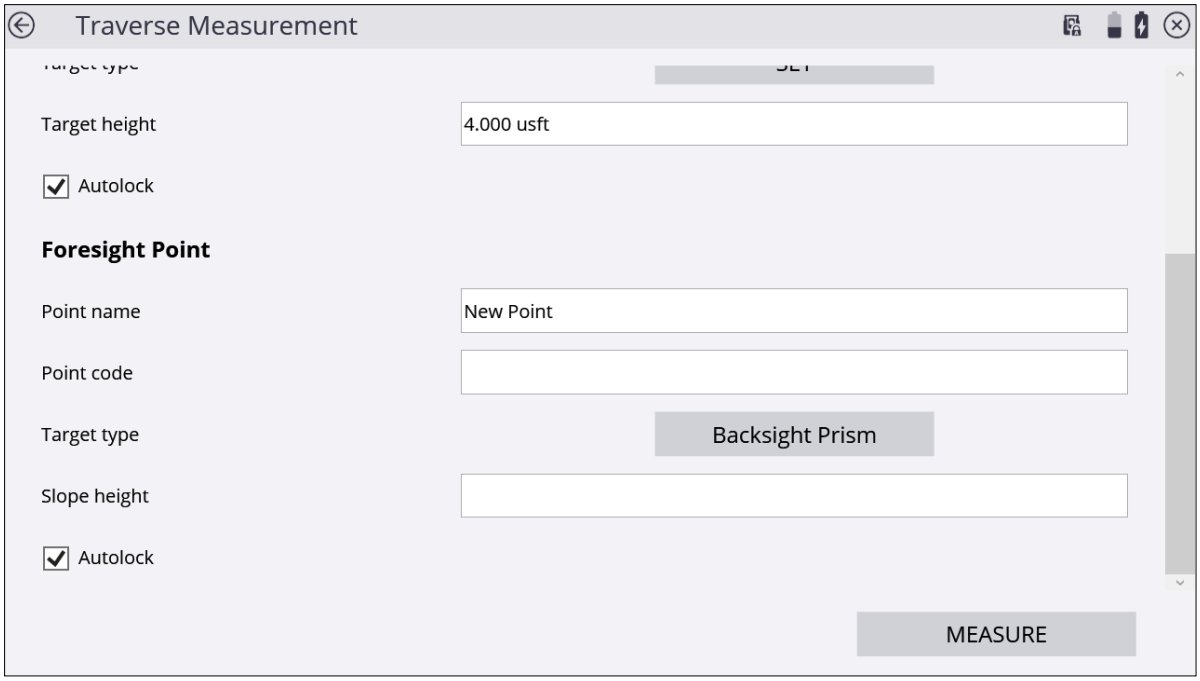
   i. Select a control point/point from the map .

   ii. Select a control point/point from the point list .

   NOTE – You can manually enter the control point/point name.

   For subsequent points, the Point name, Target type, and Target height are pre-filled using the previous instrument point information.

b. Foresight point:



When foresighting to an intermediate station, enter the point name, point code, target type, and target height.

When foresighting to the ending station, a known control point or point must be selected. Enter the target height and ensure that the point name, point code, and target type are correct.

7. After all backsight and foresight information have been verified, tap Measure.

Notes –

The first foresight point is assumed to be a new point, and its name must be entered. For subsequent foresight points at intermediate stations, the Point Type must be selected as either a New Point or Existing Point.

When checking Autolock, the instrument will lock onto the prism automatically after the first measurement set. If this option is left unchecked, the total station must be manually aimed at the target each time.

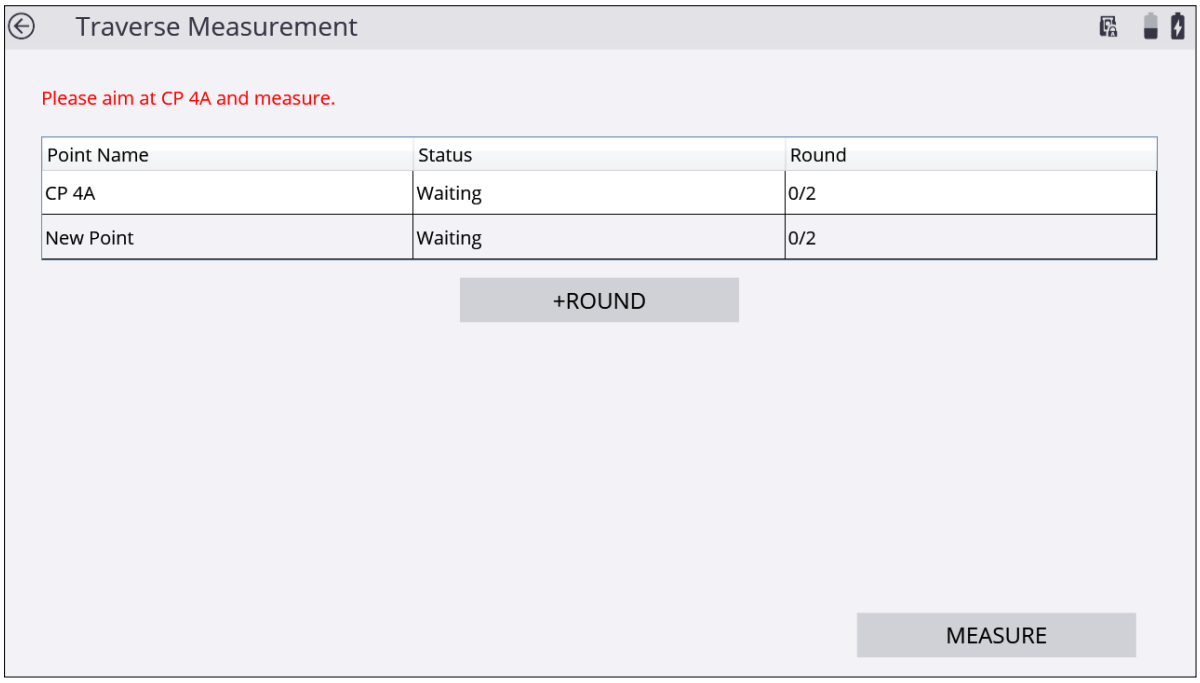
When using the Angle only option, decide if an individual control point should only be measured by angle only.

The target type and height must be entered unless using the Angle only option.

Ensure that all input values are correct because it is not possible to edit keyed-in data without re-measuring the station.

To change the Traverse Measurements Settings, tap ***Settings****. For a detailed description of the configuration options, see* *Measuring rounds of angles, page 196**.*

8. In the **Traverse Measurement** screen:
   a. Follow aiming instructions and when prompted, tap **Measure**.
   b. When all rounds of measurements are completed, tap **Next**.



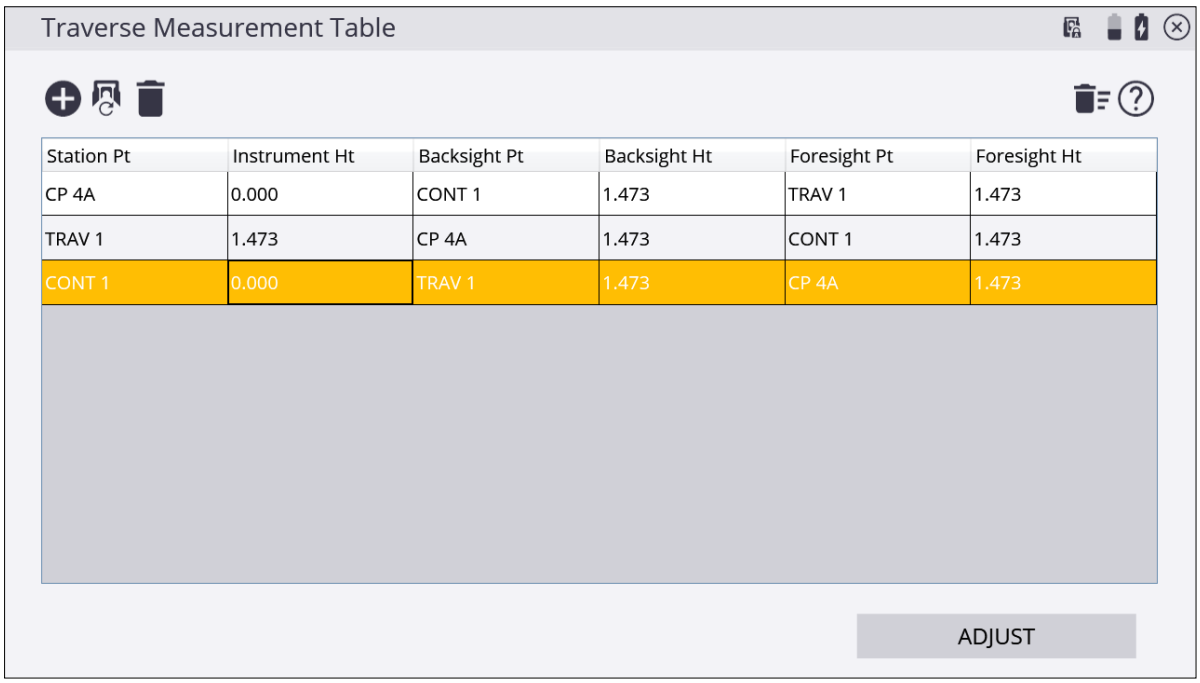
9. Repeat Step 3 through Step 7 to create traverse measurements at all required stations and complete building the traverse.

Adjusting traverses

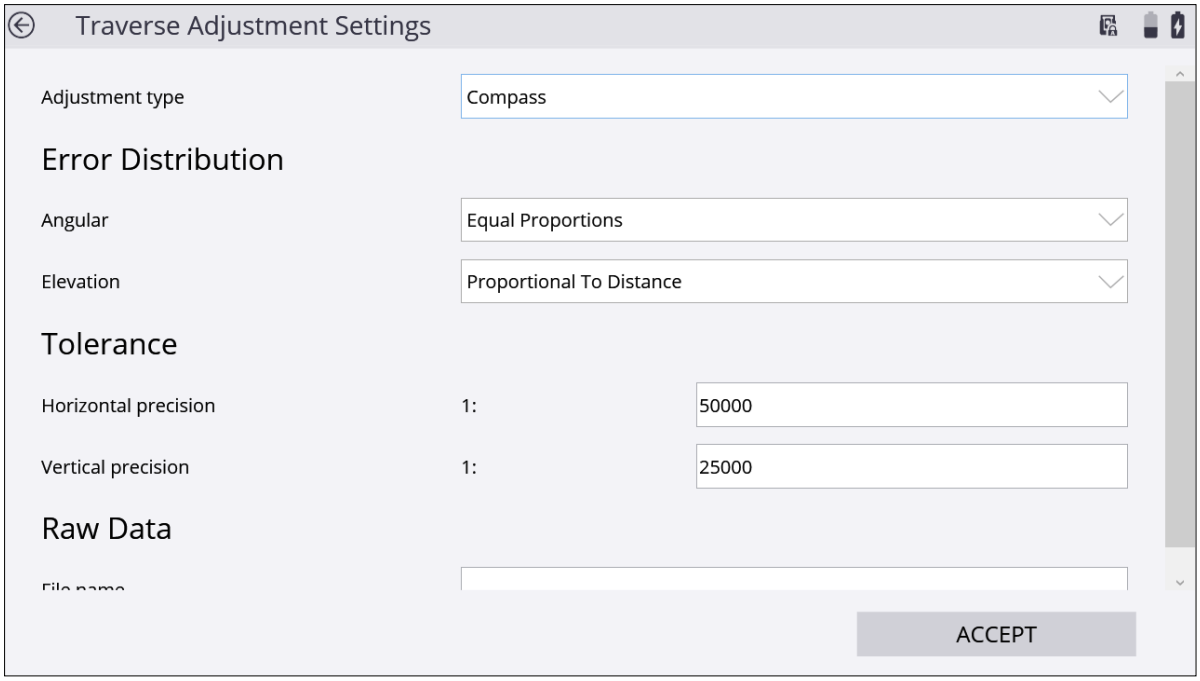
The Siteworks software will only adjust complete, closed traverses that have starting and ending stations on sets of existing control points.

To perform an adjustment:

1. After completing all traverse measurements on the **Traverse Measurement Table** screen, tap **Adjust**.



The following screen appears:



2. The adjustment settings are:
   a. **Adjustment type**: Compass or Transit.
   b. **Error Distribution** for Angular and Elevation: Proportional to Distance, Equal Proportions, or None.
   c. **Tolerance** settings for Horizontal and Vertical precisions: Enter manually.
3. When all of the adjustment settings are entered and a filename has been entered, tap **ACCEPT**.
4. After performing the adjustment calculations, the results are displayed for review in a Traverse Adjustment report. To modify the Traverse Adjustment Settings again, tap **Back**.
5. If the report generated is accepted, the adjusted traverse points can be saved as control points by tapping **ACCEPT**.

15

Utility Locator Workflow

- Pairing and connecting to Vivax-Metrotech locators
- Pairing and connecting to Radiodetection instruments
- Testing the locator in Siteworks
- Operating the locator with Siteworks

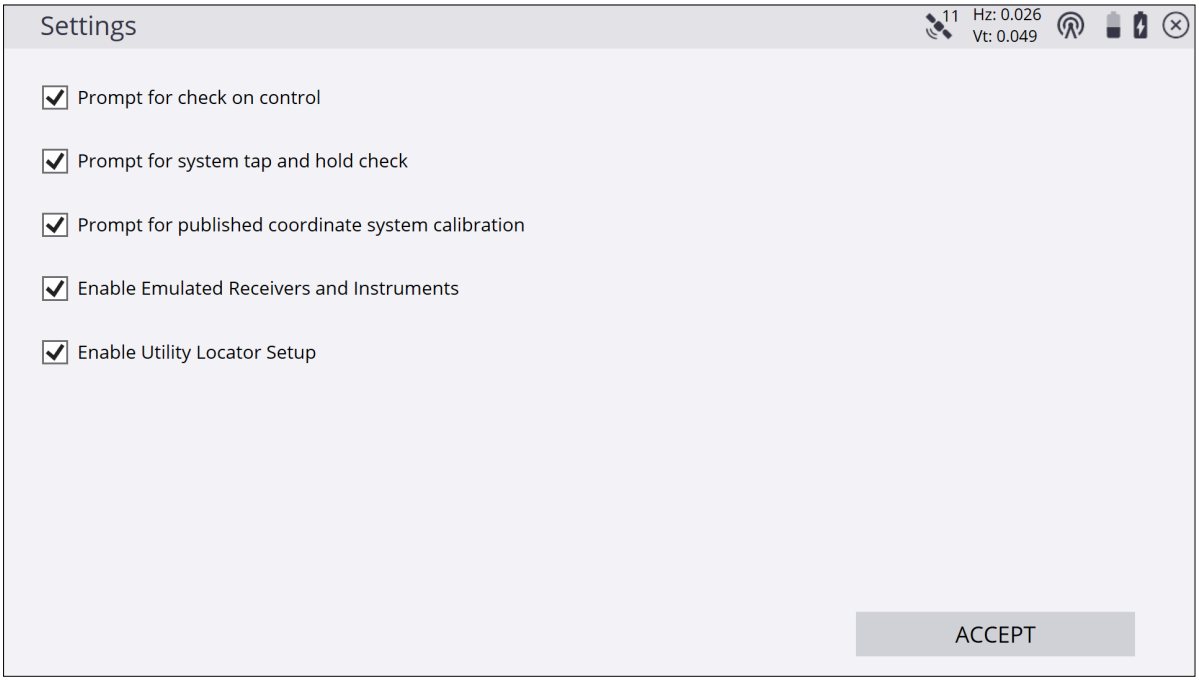
Overview

Siteworks can connect to and log the data stream of several radio frequency utility locator instruments including the Radiodetection RD8000 and 8100 models, and the Vivax-Metrotech vLocPro2 model.

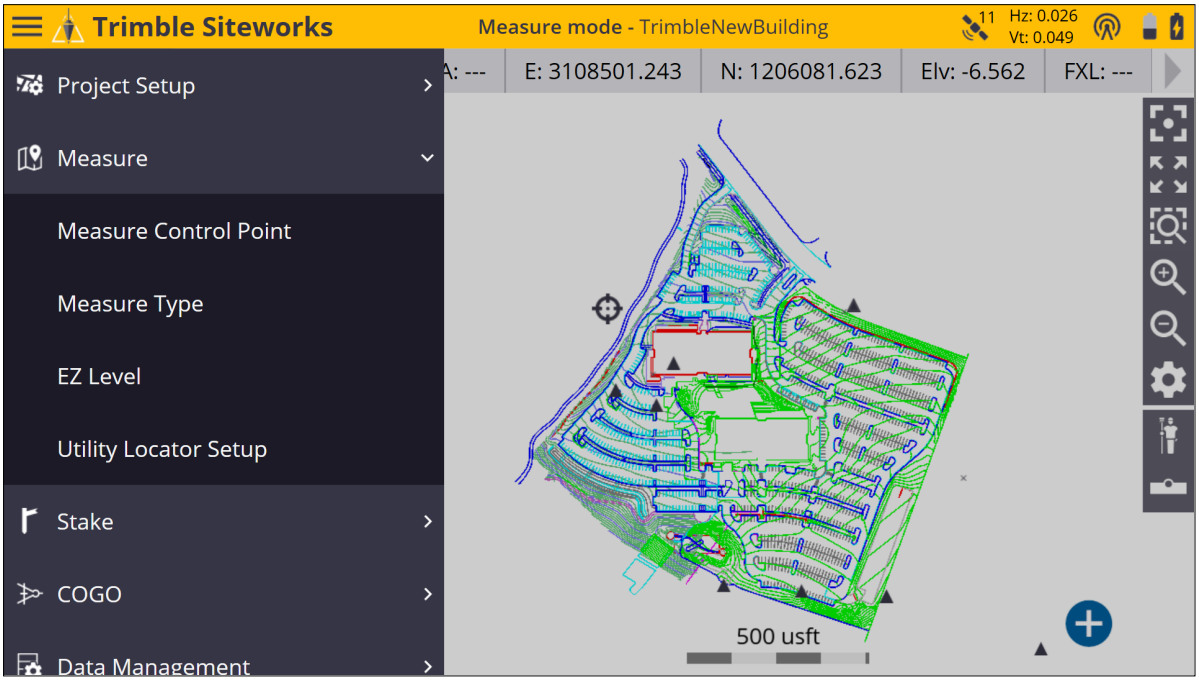
Functional integration of the locators into the Siteworks software includes the ability to record position measurements within Siteworks from GNSS and total stations and log the depth and other data output from the locator by pressing the **Depth** button on the locator wand. The locator triggers a measurement in Siteworks and also records the data streamed out of the locator by using a Bluetooth connection between the locator and the data collector.

The depth of the utility as calculated by the locator can be subtracted off the elevation at the survey rod tip. This results in the recording of the elevation of the utility. The additional information stored by Siteworks depends on the locator model, and can include, the selected locator frequency, gain settings, locate current, and other information as output by the locator. Please refer to the utility locator manuals for correct operating instructions of those devices, for additional information on setting up the streaming outputs, and the types of information output by the locators.

Locator functionality in Siteworks is enabled by selecting the **Enable Utility Locator Setup** in the CTRL+O menu.



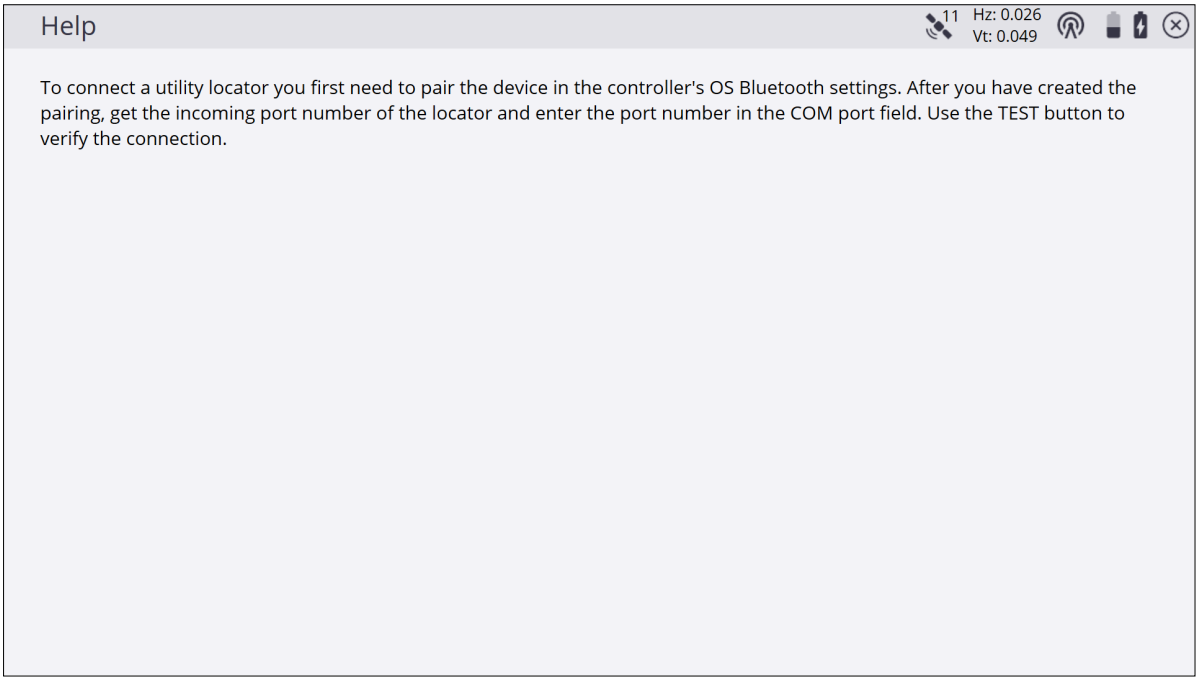
After the **Enable Utility Locator Setup** check box is selected, tapping the **Home** menu, **Measure** shows a new menu item for utility locator setup.



Tapping the **Utility Locator Setup** tab opens the **Utility Locator Setup** screen where the locator model, communication protocols, option for subtracting out the depth calculated by the locator are shown, in addition to enabling the connecting, disconnecting, and testing of the locator's output in Siteworks.

Pairing and connecting to Radiodetection instruments

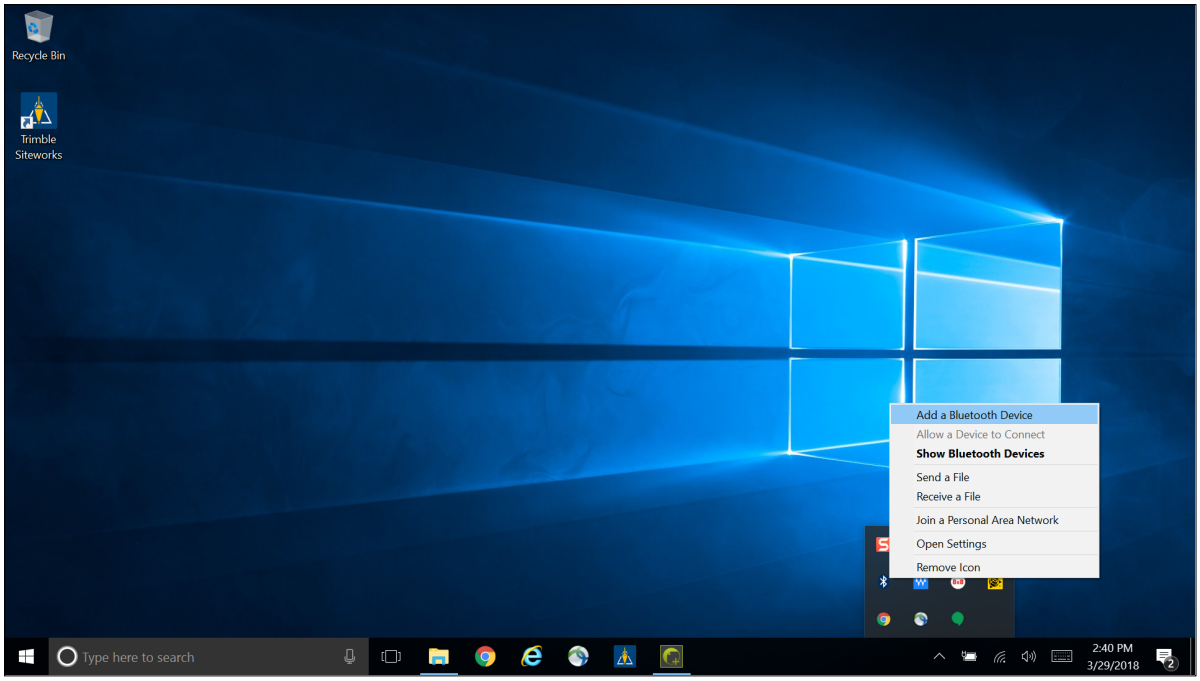
Radiodetection locators must first be paired in the data collector's Windows operating system outside of the Siteworks software. The COM port that is determined is set up by the collector in order to operate. Tapping  to the COM port field when the Radiodetection is selected in the Utility Locator Setup screen shows the following window:

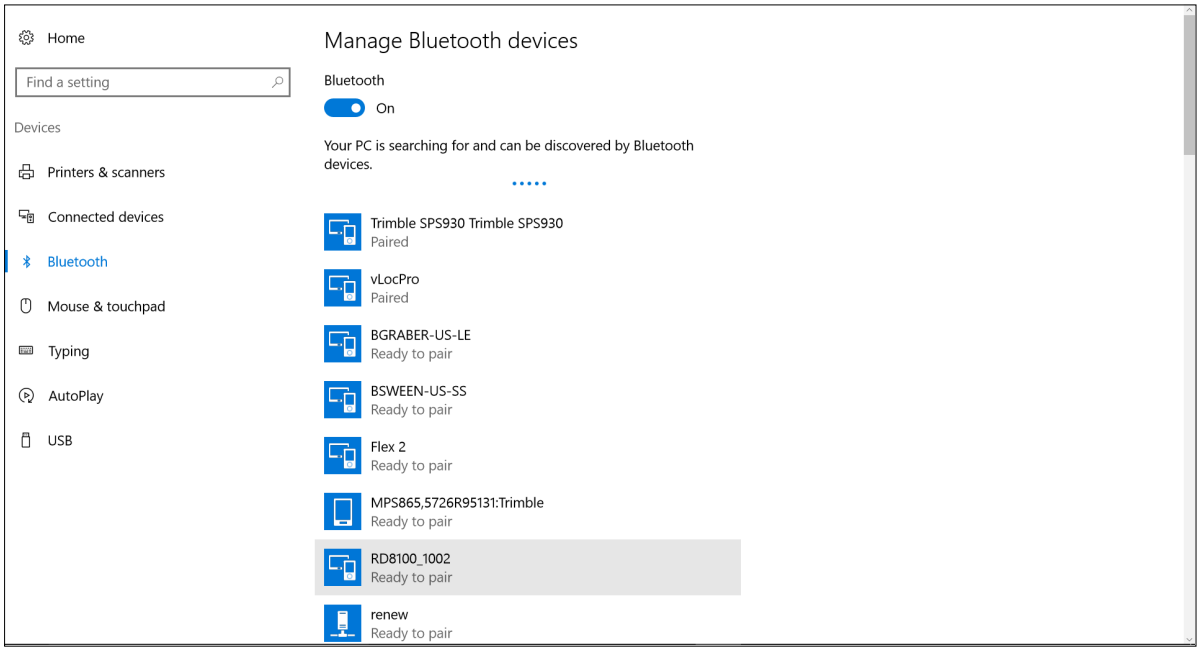


Pairing Radiodetection locators with a Windows 10 Tablet

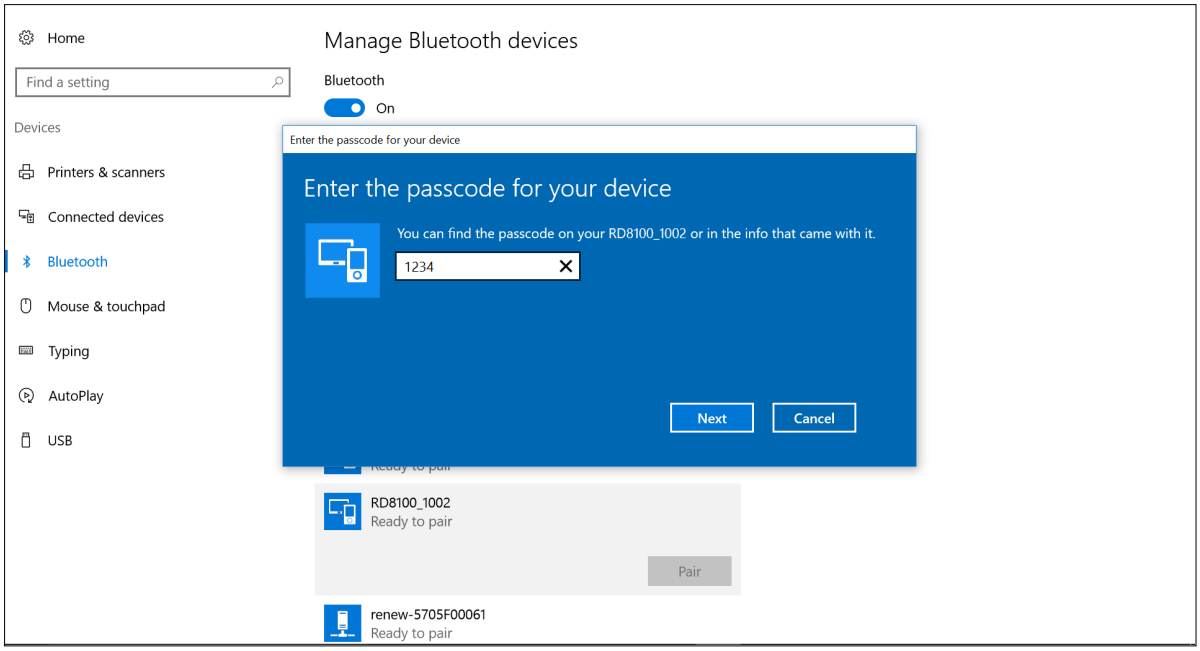
The following sections are for pairing the Radiodection instrument on the Trimble Site Tablet 10, TSC7, T10 and other Windows 10 devices.

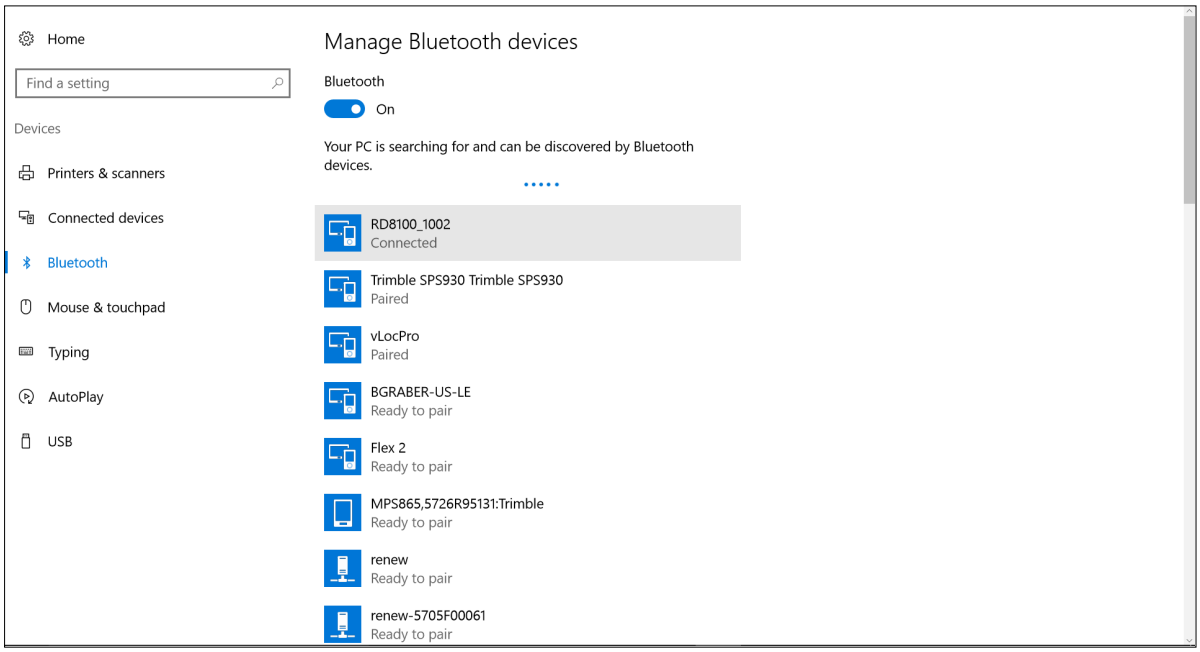
1. Select the Add a Bluetooth device from the Bluetooth menu item in the Windows taskbar and set the Radiodetection Locator into pair mode as described above.



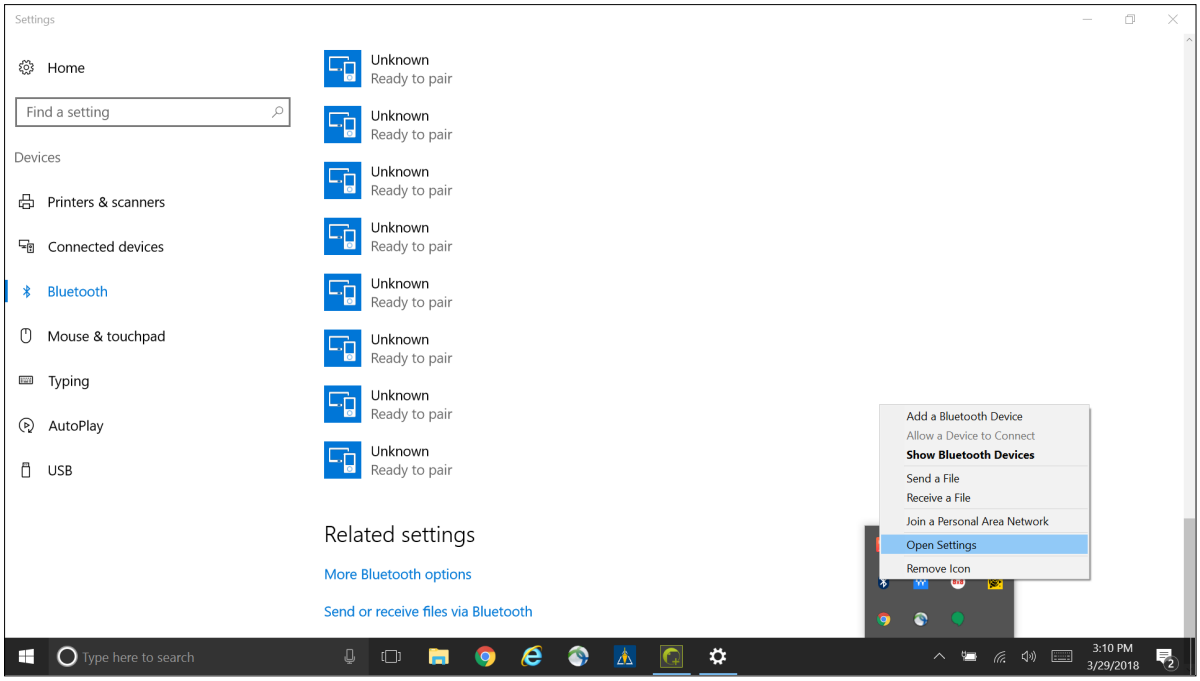


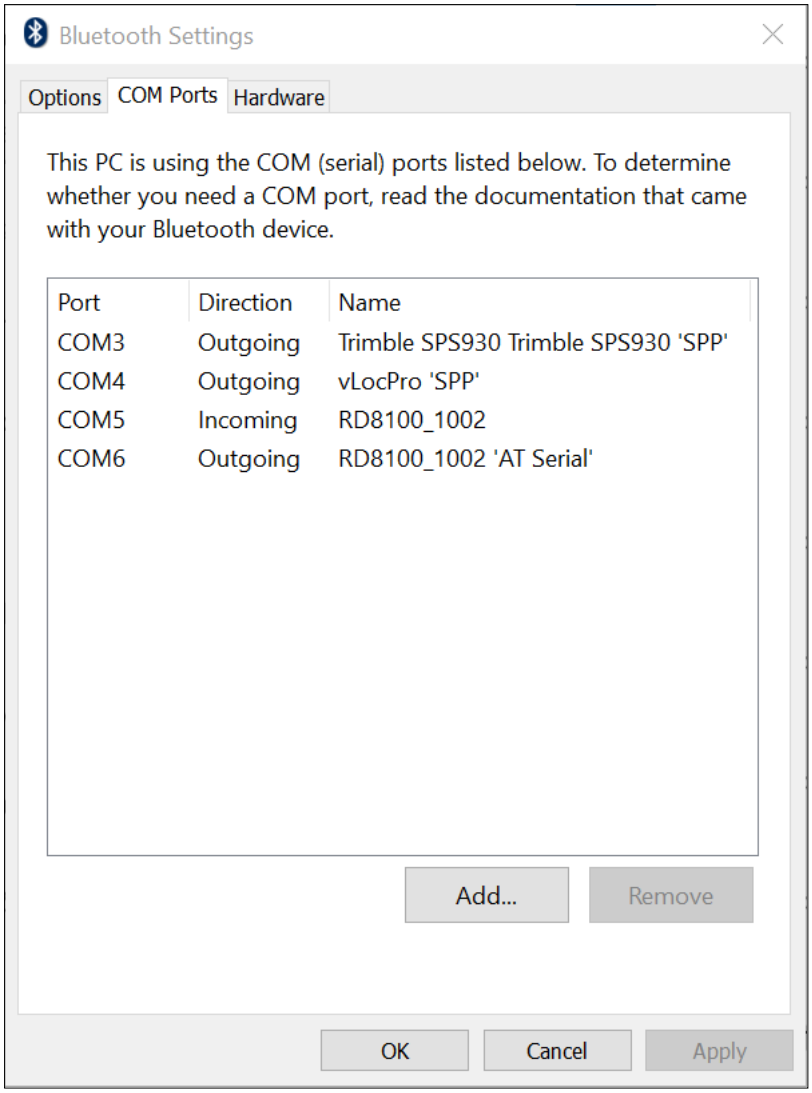
2. After the Radiodetection locator appears in the device list, click on it to begin the pairing process. Enter 1234 for a pairing code and tap **Next**.



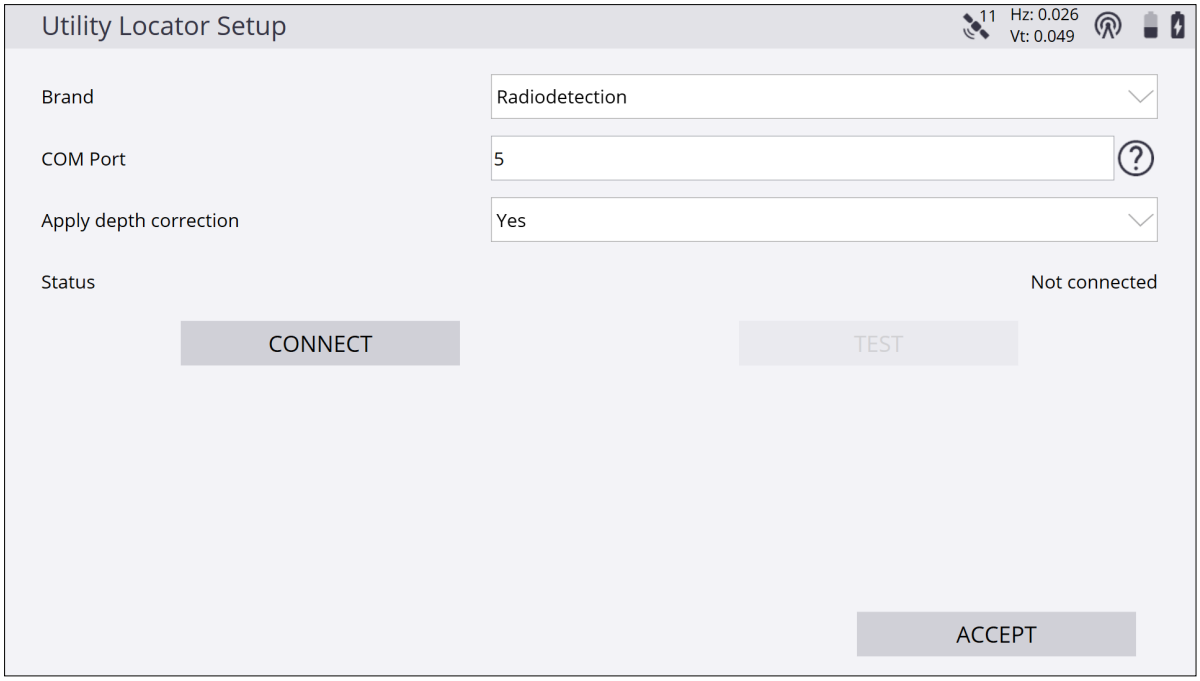


3. Determine the *incoming COM port* established for the locator by selecting the **Open Settings** command in the Bluetooth Windows taskbar icon menu. Then tap on the **COM Ports** tab to see a list of the COM ports established on the tablet:

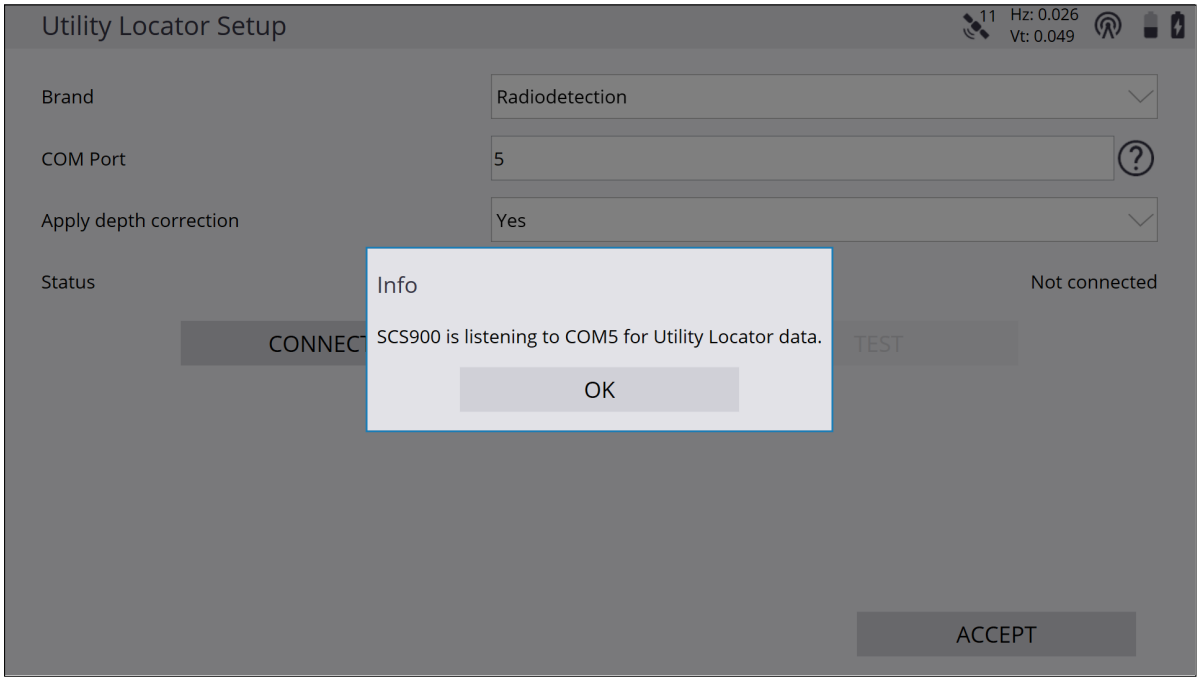

Settings
Home
Find a setting
Devices
Printers & scanners
Connected devices
Bluetooth
Mouse & touchpad
Typing
AutoPlay
USB
Unknown
Ready to pair
Related settings
More Bluetooth options
Send or receive files via Bluetooth
Add a Bluetooth Device
Allow a Device to Connect
Show Bluetooth Devices
Send a File
Receive a File
Join a Personal Area Network
Open Settings
Remove Icon
Type here to search
3:10 PM
3/29/2018



4. Note the incoming port assigned to the RD8100 or RD8000 displayed in the window (in this example COM5) as this will be entered into the Siteworks software.
5. Back in the Siteworks software, return to the **Utility Locator Setup** screen. From the **Brand** field, select Radiodetection. Enter the incoming COM port determined in the above steps during Bluetooth pairing.



6. After configuring the device, entering the COM port, and selecting if the calculated depth from the locator is to be subtracted from the measured surface elevation, tap **Connect** to connect to the locator.



Pairing and connecting to Vivax-Metrotech locators

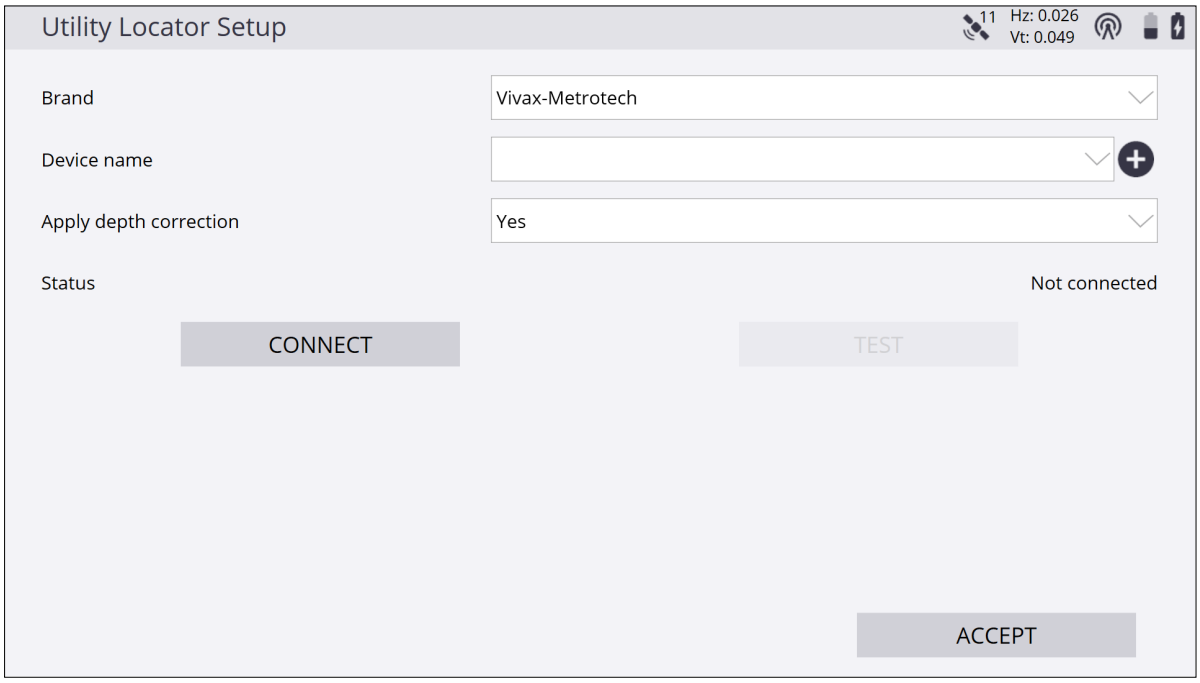
For Vivax instruments it is only necessary to ensure that the Bluetooth Search setting in the settings menu on the locator is set to "Enabled" to establish a connection with Siteworks.

To access the locator's settings menu:

1. Press and hold the "**i**" (information depth/current) button on the Locator, and then the "**+**" button on the locator to navigate to the locator's Bluetooth menu options.
2. Press the Locate Mode (rightmost) button on the locator to enable the Bluetooth search setting until the screen on the locator displays the message: **Bluetooth Search: Enabled**.
3. Once this is enabled, you can perform a Bluetooth search and connect to the locator from within the Siteworks **Utility Locator Setup** window.

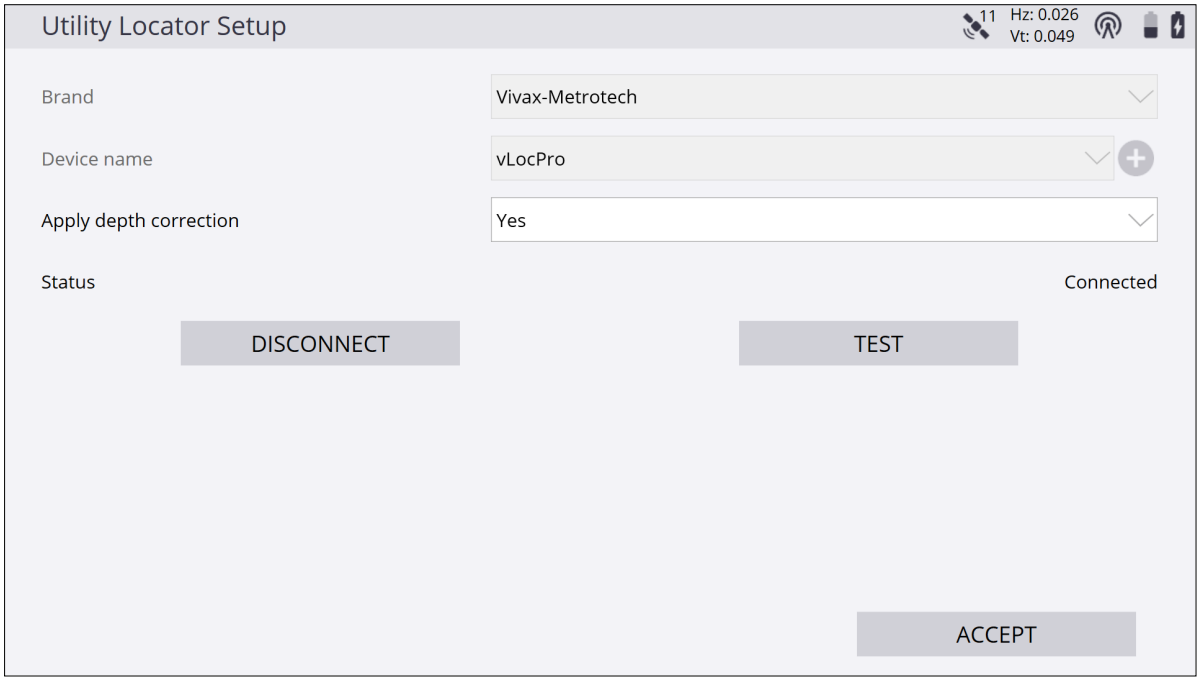
To connect to the vLocPro2:

1. From the **Brand** field, select Vivax-Metrotech.
2. On the right side of the **Device name** field, tap  to begin a Bluetooth search for the Locator in Siteworks:



3. After finding the Locator and completing the Bluetooth pairing process within Siteworks, select the vLocPro in the **Device name** field.

4. Set the required option for applying the depth correction to measured points and then tap **Connect**.
5. The Bluetooth icon on the Locator screen should turn blue and illuminate and the **Connect** button should change to **Disconnect**, indicating a connection has been made to Siteworks:

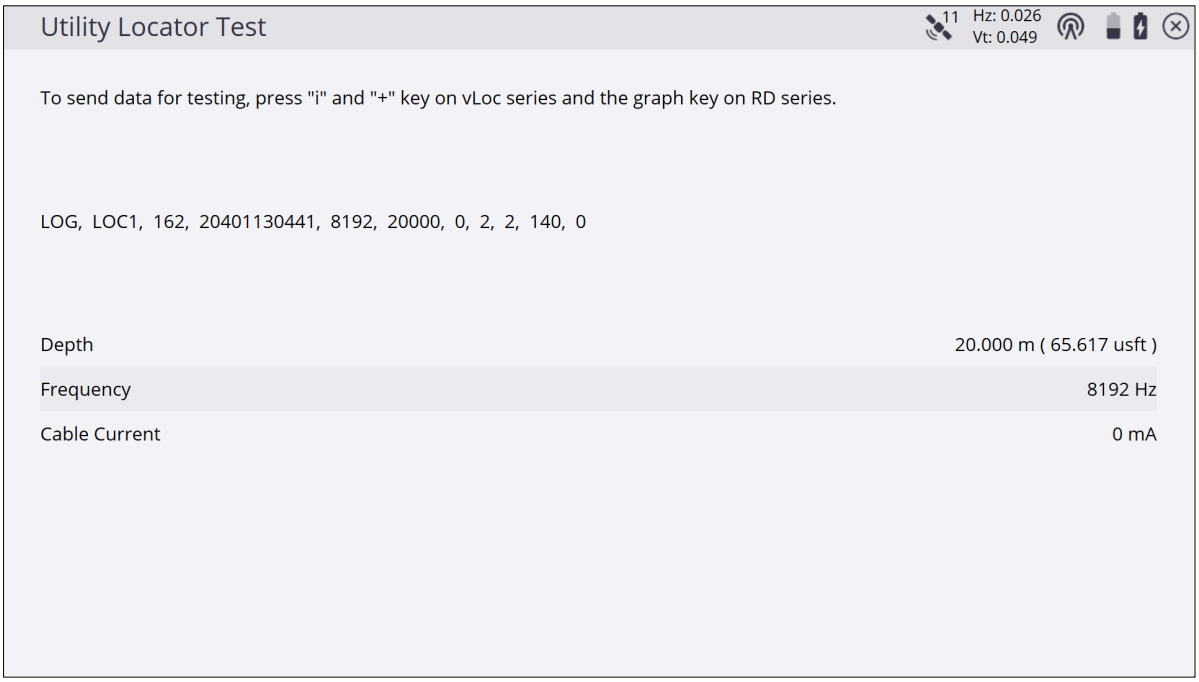


Testing the locator in Siteworks

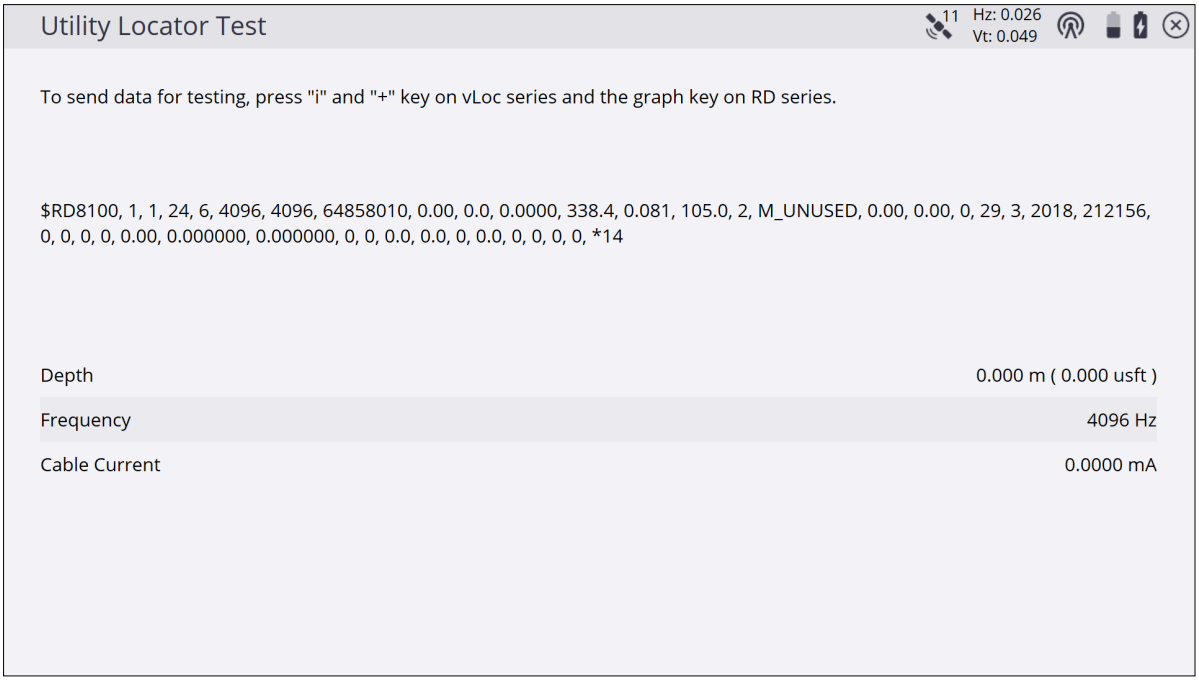
After pairing and connecting the locator in the Siteworks software, pressing the **Test** button in the **Setup** screen will open a screen where the output from the locator can be seen. This screen enables the checking of the connection between the collector and the locator and previewing the data that will be logged by Siteworks.

Press the **Depth** button on the Radiodetection devices or the "**i**" and then "**+**" button on Vivax-Metrotech devices to send the string of data from the locator to Siteworks and ensure that the correct data are being sent. Tap **Close** when done to return to the **Utility Locator Setup** Screen.

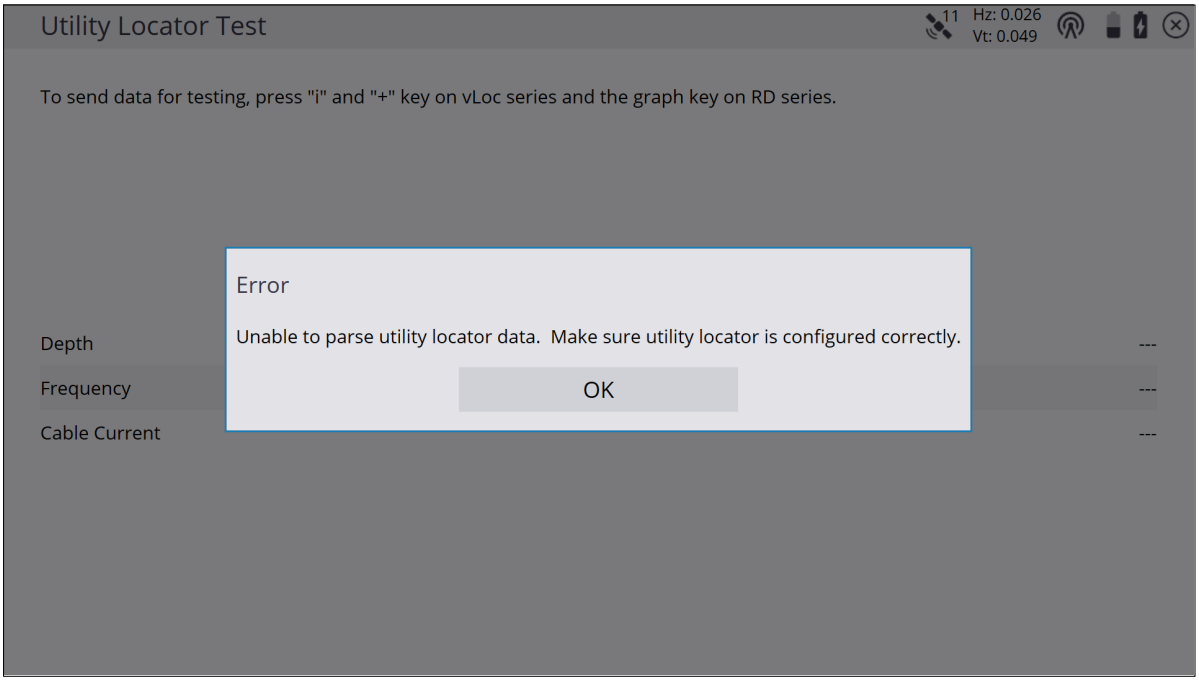
Example Vivax-Metrotech test data:



Example Radiodetection test data:



Note that if you are using a Radiodetection locator and receive the error shown below, this indicates that the locator's output is not configured to the ASCII – 1 protocol. See Pairing and connecting to Radiodetection instruments, page 215 for how to configure Radiodetection Locators to the correct output protocol.



Operating the locator with Siteworks

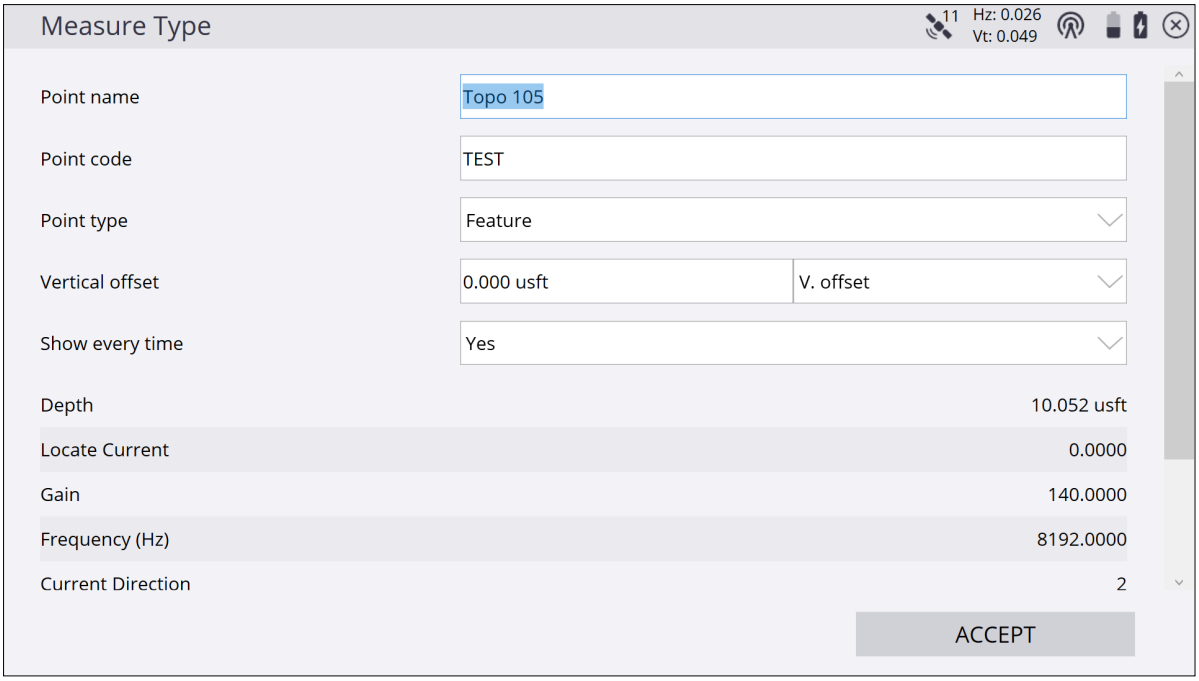
Once the locator is correctly configured and connected to the Siteworks software, whenever Siteworks is in measure mode pressing the **depth** button on the locator will trigger either a point or line measurement in Siteworks depending on what measure type is selected. Depth and other data streamed from the location is also recorded.

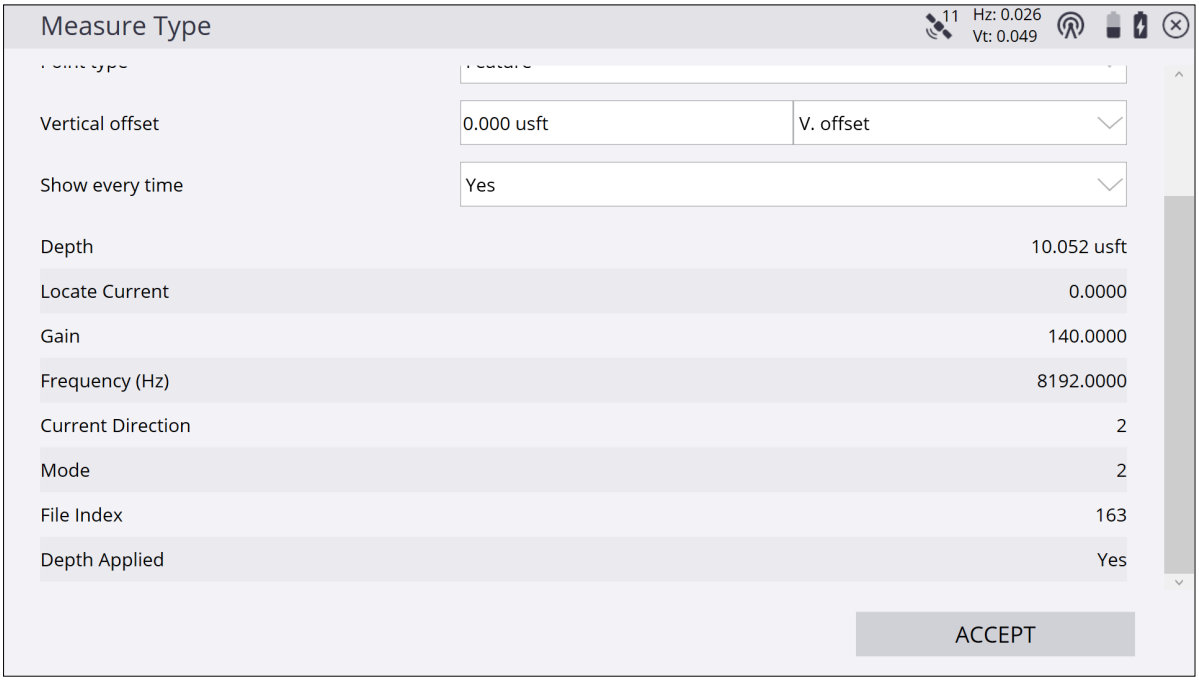
Pressing the **Measure** key while connected to a locator will record a normal measurement that will not include any information from the locator. The only way to record locator data in Siteworks is by triggering the measurement by pressing the **Depth** button on the locator. The locator cannot be used to trigger a measurement while in Stake mode.

If the **Apply Depth Correction** is set to Yes in the **Utility Locator Setup** screen, then the depth calculated by the locator will be subtracted from the survey rod tip elevation. This will record the elevation of the utility. The original, unadjusted surface elevation is also recorded with the point, but the measured point stored in Siteworks will represent the surface elevation at the rod tip minus the depth the locator calculated.

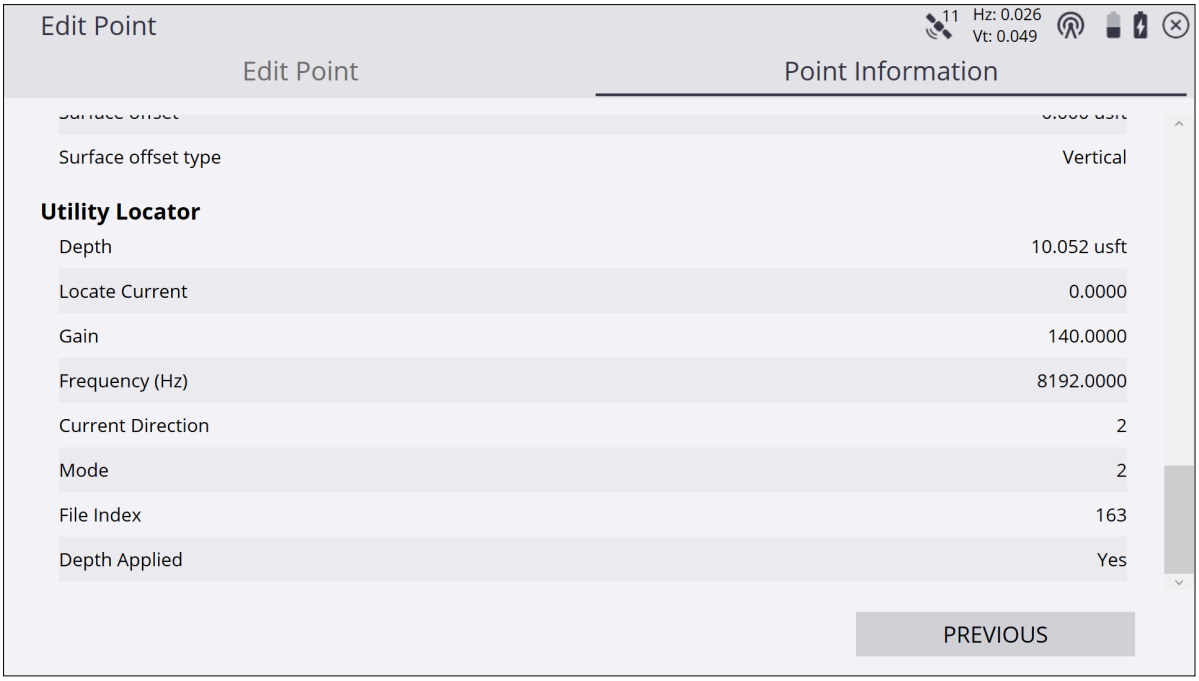
For best results, ensure that both the locator and survey rod tip are in the proper location when a measurement is triggered with the locator depth button. Refer to the locator's operating instructions for how to obtain the best quality depth readings, to understand the limitations of the calculated depth values, and for more information on setting up the Bluetooth connections.

A preview of the data output from the locator is shown in the **Measure Type** screen for point measurements before being stored. Tap **Accept** to record the point and store the data. This preview function is not available for line measurements, as those are always stored without a preview option.





The stored locator data for both points and line points can also be reviewed by tapping and holding on a point and selecting **Point Information** or via the Point Manager.



While measuring, consider the line measurement type, especially if applying depth corrections based on the locator output. For example, if you are measuring surface topography for grade checking, then it may not be desired to measure utility lines as breaklines or volume boundaries, or include utility point measurements in as surface points (i.e., set them as "feature" points in the **Measure Type** settings) as not to disrupt any other measured surface points.

To export the recorded locator data for use in other software programs, select the **Include QA data** option in the CSV export of Measured Data. The locator data is also included in the exported Record.txt file.

Glossary

benching	Benching is a process of aligning your GNSS position (latitude, longitude, and height) to a benchmark that has been added as a reference point. The calibration offsets your GNSS position to that of the benchmark, improving accuracy and providing a point that you can return to later.
AutoBase	AutoBase technology uses the position of the receiver to automatically select the correct base station; allowing for one button press operation of a base station. It shortens setup time associated with repeated daily base station setups at the same location on jobsites.
BaseAnywhere	BaseAnywhere technology allows the user to place the GNSS base station anywhere on site and does not require the base station to be setup on an existing control point. After the base is configured in BaseAnywhere mode it calculates an autonomous position and begins broadcasting corrections via radio or Wi-Fi. The rover is then used to bench into a control point which calculates the necessary offsets and parameters to allow for fully accurate RTK GNSS operations.
base station	Also called *reference station*. In construction, a base station is a receiver placed at a known point on a jobsite that tracks the same satellites as an RTK rover, and provides a real-time differential correction message stream through radio to the rover, to obtain centimeter level positions on a continuous real-time basis. A base station can also be a part of a virtual reference station network, or a location at which GNSS observations are collected over a period of time, for subsequent postprocessing to obtain the most accurate position for the location.
DGPS	See real-time differential GPS.
design map	The map that provides live linework within a design for stakeout operations. The design map is a DXF file.
differential correction	Differential correction is the process of correcting GNSS data collected on a rover with data collected simultaneously at a base station. Because the base station is on a known location, any errors in data collected at the base station can be measured, and the necessary corrections applied to the rover data. Differential correction can be done in real-time, or after the data is

	collected by postprocessing.
differential GPS	See real-time differential GPS.
elevation (elev, elv)	(1) Vertical distance (height) above or below mean sea level. (2) Vertical distance above or below the geoid. (3) Distance above or below Local Datum.
elevation mask	The angle below which the receiver will not track satellites. Normally set to 10 degrees to avoid interference problems caused by buildings and trees, atmospheric issues, and multipath errors.
feature	A feature is a physical object or event that has a location in the real world, which you want to collect position and/or descriptive information (attributes) about. Features can be classified as surface or non-surface features, and again as points, lines/breaklines, or boundaries/areas.
GLONASS	Global Orbiting Navigation Satellite System. GLONASS is a Soviet space-based navigation system comparable to the American GPS system. The operational system consists of 21 operational and 3 non-operational satellites in 3 orbit planes.
GNSS	Global Navigation Satellite System.
GPS	Global Positioning System. GPS is a space-based satellite navigation system consisting of multiple satellites in six orbit planes.
height	It can mean a target height or antenna height (for example, 2 m of rod height).
Here position	An autonomous instantaneous position derived from the GPS receiver's uncorrected latitude, longitude, and height.
IBSS	Internet Base Station Service. This Trimble service makes the setup of an Internet-capable receiver as simple as possible. The base station can be connected to the Internet (cable or wirelessly). To access the distribution server, the user enters a password into the receiver. To use the server, the user must have a Trimble Connected Community site license.
Location GPS	Location GPS covers decimeter to submeter GNSS positioning techniques including Satellite Based Augmentation Systems (SBAS) such as WAAS, EGNOS, and MSAS), DGPS (reference station and rover operations), OmniSTAR VBS/HP/XP services, and Location RTK (decimeter-level RTK positioning).

Location RTK	Some applications such as vehicular-mounted site supervisor systems do not require Precision RTK accuracy. Location RTK is a mode in which, once initialized, the receiver will operate either in 10 cm horizontal and 10 cm vertical accuracy, or in 10 cm horizontal and 2 cm vertical accuracy.
Precision GPS	GPS positioning provided by techniques that typically deliver centimeter-level accuracy. These include RTK (Real-Time Kinematic) techniques and signals obtained from a VRS (Virtual Reference Station) system.
Project	A project that is to be worked on for a significant period of time. A project stores all design data and all executed work orders so you can easily find data whether you are in the office or in the field.
postprocessing	Postprocessing is the processing of satellite data after it is collected, in order to eliminate error. This involves using computer software to compare data from the rover with data collected at the base station.
real-time differential GPS	Also known as *real-time differential correction* or *DGPS*. Real-time differential GPS is the process of correcting GPS data as you collect it. Corrections are calculated at a base station and then sent to the receiver through a radio link. As the rover receives the position it applies the corrections to give you a very accurate position in the field. Most real-time differential correction methods apply corrections to code phase positions. While DGPS is a generic term, its common interpretation is that it entails the use of single-frequency code phase data sent from a GNSS base station to a rover GNSS receiver to provide sub-meter position accuracy. The rover receiver can be at a long range (greater than 100 kms (62 miles)) from the base station.
Road job	A road job is the term that defines a complete road model within the Terramodel and Siteworks software. It is a collection of roadway information that is expected to function together to define a roadway or a portion of a roadway between specific stationing limits. A road job contains the main alignment and all sub alignments, the road templates, and all the information used to define widening and super elevation for the road. A single project can contain multiple road jobs for different roads also contained within that single construction project.
road model	The road model used by the Siteworks software is a Trimble

	Terramodel PRO file. This file can be used for both staking and grade checking operations. The Road model is a template based model that provides full accuracy anywhere within the roadway surface.
rover receiver	A rover is any mobile GNSS receiver that is used to collect or update data in the field, typically at an unknown location.
Roving mode	Roving mode applies to the use of a rover receiver to collect data, stakeout, or control earthmoving machinery in real time using RTK techniques.
RTK	real-time kinematic. A real-time differential GPS method that uses carrier phase measurements for greater accuracy.
site map	The site map within the Siteworks software is stored as a part of the site data. The site map provides linework as a reference only and is not live, which is why you cannot select it for stakeout purposes.
station	A station is the running distance along the centerline or road that starts at 0.0 and increments as you proceed along the route. This term is used primarily in the US, whereas the equivalent term chainage is used throughout many other areas of the world, such as Australia, Asia, Europe, and New Zealand.
surface model	The surface model used by the Siteworks software is a Trimble Terrain Model file (TTM file). It provides a 3D surface model that can be used for stakeout or grade checking operations.
work order	A work order covers a task to be performed by a crew on a single jobsite. A work order contains the reference to the appropriate design, required settings and tolerances for the task, and a record and report of all the data measured or staked out in the process of completing the task. A work order can cover a short-term task (such as the stakeout of a specific building pad) or a task that will last the duration of the project (such as the stakeout of storm water drainage) and that will be executed periodically as required during the project. When the project is complete, all the information regarding the task is stored in a single file that is easy to recall.