MAGNET Field Site Help

Version: 3.0

Part Number 1001627-01

Revision D

March, 2015

Software End User License Agreement

IMPORTANT: PLEASE READ CAREFULLY. The software product provided to you by Topcon Positioning Systems, Inc. ("TPS") along with its associated manuals and documentation (collectively, the “Software”) is owned by TPS and your use is subject to the terms and conditions of this Software License End-User Agreement (“Agreement”) set forth below. If you are entering into this agreement on behalf of a company or other legal entity, you represent that you have the authority to bind such entity to these terms and conditions, in which case the terms "you" or "your" shall refer to such entity. If you do not have such authority, or if you do not agree with these terms and conditions, you may not use the service. TPS also reserves the right to immediately terminate this agreement and suspend, cancel or delay the service for failure to comply with the terms provided herein.

By clicking the “ACCEPT” button below, or by installing or using the Software, you agree to be bound by the terms and conditions of this Agreement. In addition, by clicking the “ACCEPT” button below, you agree to be bound by the terms and conditions of the Terms of Use of the Web site www.magnet-enterprise.com (the “TPS Site”). If you do not agree, exit out of the TPS Site and you will not be authorized to use the Software or otherwise utilize the documentation accompanying the Software.

Safety. IMPROPER USE OF A TOPCON PRODUCT CAN LEAD TO INJURY TO PERSONS OR PROPERTY AND/OR MALFUNCTION OF THE PRODUCT. THE PRODUCT SHOULD ONLY BE REPAIRED BY AUTHORIZED TPS WARRANTY SERVICE CENTERS. USERS SHOULD REVIEW AND HEED THE SAFETY WARNINGS IN THE MANUAL ACCOMPANYING THE PRODUCT.

Ownership of the Software. The Software and the accompanying documentation are owned by TPS and its respective licensors and are protected by United States and international copyright laws and other intellectual property laws.

Professional Use. The Software is designed to be used by a professional. The user is required to be a professional surveyor or have a good knowledge of surveying and be familiar with the safe use of such products, in order to understand the user instructions before operating the Software.

Restrictions on Use and Transfer. You may not modify, adapt, translate, reverse engineer, decompile, or disassemble the Software or create derivative works from the Software, any component thereof or any related documentation, nor may you remove, modify or hide or otherwise make unreadable or non-viewable any notice, legend, advice, watermark trademark, service mark, or other designation contained on the Software, component thereof, documentation, or output therefrom. You may not distribute registered copies of the Software to third parties, including, without limitation rent, lease, or lend the Software to third parties. You agree to not use, permit the use, or use the Software in violation of any U.S. Federal, state, or local laws or regulations or any foreign law or regulation, including laws regarding intellectual property rights in or laws or regulations regarding the trading or exchange of securities or concerning the Software. You further agree to use the Software solely for its intended purpose. The Software and all related information you may come to learn related to the Software and TPS’ operations in connection therewith is confidential in nature. You agree to take all reasonably necessary precautions to protect TPS' confidential information and exercise at least the same degree of care in safeguarding the confidential information as you would with your own most valuable confidential information.

Support Services. TPS may provide you with support services related to the Software ("Support Services"). Any supplemental software code provided to you as part of the Support Services shall be considered part of the Software and subject to the terms and conditions of this Agreement.

Grant of License to Software. If you have purchased or otherwise received Software from TPS, TPS grants you the right to install and use copies of the Software on your computer running a validly licensed copy of the operating system for which the Software was designed (e.g., Windows CE 6.0, Windows Mobile, Windows XP, Windows Vista, Windows 7). This license is coterminous with the Term set forth in the Master Subscription Agreement for the Magnet Terms of Use and is a personal, non-exclusive, non-transferable (except as expressly set forth herein) license to use such Software under the terms stated herein and in any case only with a single Device. A "Device" means a personal computer or the product upon which the Software is intended (pursuant to its applicable documentation) to be installed and used. You may permanently transfer rights under this Agreement only as part of a permanent sale or transfer of the Device, and only if the recipient agrees to this Agreement. If the Software is an upgrade, any transfer must also include all prior versions of the Software. The license is effective until terminated. You may terminate the license at any time by destroying the Software and related documentation. Without prejudice to any other rights of TPS, TPS may terminate your license if you fail to comply with the terms and conditions of this Agreement. In such event, you must destroy all copies of the Software in your possession.

Maintenance of the Software. TPS agrees to provide Maintenance (as defined herein) to you for the first year from the date you enter into this Agreement, at no additional charge, pursuant to the terms and conditions set forth herein. Following the first year, and for successive years thereafter ("each year defined as a Term(s)"), in order to receive continued Maintenance, you must pay the Maintenance Fee at the start of each Term as set forth by TPS. Failure to renew Maintenance for a new Term may result in you having to enter into a new Agreement in order to receive Maintenance.

Maintenance is defined as any improvements or modifications to the Software that TPS makes generally available. Any such improvements or modifications shall become part of the Software for all purposes of this Agreement. You acknowledge and agree that the Maintenance to be provided by TPS hereunder is limited to the most current version of the Software and the immediately preceding version. Activation of your Software is required in order to receive the most currently available Maintenance features.

Disclaimers. EXCEPT FOR ANY WARRANTIES IN A WARRANTY CARD ACCOMPANYING THE SOFTWARE (AND ALL MAINTENANCE), THE SOFTWARE IS PROVIDED "AS IS." TPS MAKES NO PROMISES, REPRESENTATIONS OR WARRANTIES, WHETHER EXPRESSED OR IMPLIED, REGARDING OR RELATING TO THE SOFTWARE (INCLUDING, WITHOUT LIMITATION, THAT THE SOFTWARE WILL BE ERROR-FREE OR AVAILABLE FOR USE AT ALL TIMES) OR CONTENT THEREIN OR TO ANY OTHER MATERIAL FURNISHED OR PROVIDED TO YOU PURSUANT TO

THIS AGREEMENT OR OTHERWISE, AND TPS SPECIFICALLY DISCLAIMS ALL IMPLIED WARRANTIES OF MERCHANTABILITY, NON-INFRINGEMENT AND FITNESS FOR A PARTICULAR PURPOSE WITH RESPECT TO SAID MATERIALS OR THE USE THEREOF. TO THE MAXIMUM EXTENT ALLOWABLE BY LAW, THE UNIFORM COMMERCIAL CODE OR OTHER UNIFORM LAWS SHALL NOT APPLY TO THIS AGREEMENT.

Trademarks. The TPS name, the TPS logo, and the product names associated with the Service are trademarks of TPS and no right or license is granted to use them. Product and company names mentioned herein may be trademarks of their respective owners.

Microsoft® Bing™ maps is used in Magnet™. Microsoft® Terms of Use for Bing™ maps can be found at: http://www.microsoft.com/maps/assets/docs/terms.aspx#11. © 2011 Microsoft Corporation. All rights reserved.

RealDWG ™ and Autodesk® RealDWG by Autodesk, Inc are used in Magnet™. Copyright © 1998-2011 Autodesk, Inc. All rights reserved. Information can be found at www.autodesk.com/autodeskrealdwg

Limitation of Liability. TPS AND ITS DISTRIBUTORS SHALL NOT BE LIABLE FOR TECHNICAL OR EDITORIAL ERRORS OR OMISSIONS CONTAINED IN THE SOFTWARE OR ITS DOCUMENTATION. TO THE MAXIMUM EXTENT ALLOWABLE BY LAW, IN NO EVENT SHALL TPS OR ITS PERSONNEL BE LIABLE UNDER ANY CLAIM, DEMAND OR ACTION ARISING OUT OF OR RELATING TO THE SOFTWARE, ITS USE, INSTALLATION OR TPS'S PERFORMANCE OR LACK THEREOF UNDER THIS AGREEMENT FOR ANY SPECIAL, INDIRECT, INCIDENTAL, EXEMPLARY OR CONSEQUENTIAL DAMAGES, WHETHER OR NOT TPS HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH CLAIM, DEMAND OR ACTION. UNDER NO CIRCUMSTANCES, INCLUDING BUT NOT LIMITED TO A NEGLIGENT ACT, WILL TPS OR ITS AFFILIATES, AGENTS, EMPLOYEES, OR LICENSORS BE LIABLE FOR ANY DAMAGES OF ANY KIND THAT RESULT FROM (i) THE USE OR THE INABILITY TO USE THE SOFTWARE; (ii) THE TIMELINESS, DELETION, MISDELIVERY, OR FAILURE TO STORE ANY DATA, COMMUNICATIONS OR SETTINGS; (iii) THE COST OF GETTING SUBSTITUTE GOODS AND SERVICES; OR (iv) UNAUTHORIZED ACCESS TO OR ALTERATION OF YOUR TRANSMISSIONS OR DATA EVEN IF SUCH PARTY HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. IN ADDITION, WITHOUT LIMITING THE FOREGOING, IN ALL EVENTS THE REMEDIES AVAILABLE TO YOU SHALL BE LIMITED TO THE GREATER OF THE AMOUNT PAID BY YOU TO TPS FOR THE RIGHT TO USE THE SOFTWARE OR $100.

Indemnification. YOU SHALL INDEMNIFY AND HOLD TPS AND ITS AGENTS, OFFICERS, MANAGERS, EMPLOYEES AND MEMBERS HARMLESS FROM AND AGAINST ANY AND ALL CLAIMS, DEMANDS, SUITS, JUDGMENTS AND EXPENSES (INCLUDING REASONABLE LEGAL FEES AND EXPENSES) ARISING OUT OF OR RELATING TO THE BREACH BY YOU OF YOUR OBLIGATIONS UNDER THIS AGREEMENT OR TO INTELLECTUAL PROPERTY MISAPPROPRIATION OR INFRINGEMENT CLAIMS BY ANY THIRD PARTY BASED ON OR

RELATING TO YOUR USE OF THE SOFTWARE. YOU AGREE TO GIVE TPS PROMPT NOTICE OF SUCH CLAIMS AND TO PERMIT TPS TO CONTROL THE DEFENSE OR SETTLEMENT THEREOF.

Export Restrictions. You agree with all international and national laws that apply to the Software, including the U.S. Export Administration Regulations, as well as end-user, end use and country destination restrictions issued by U.S. and other governments.

Website; Other Statements. No statement contained at the TPS Website (or any other web site) or in any other advertisements or TPS literature or made by an employee or independent contractor of TPS modifies these Terms and Conditions (including the Software License Agreement, Disclaimer of Warranty and limitation of liability).

General. The above Terms and Conditions may be amended, modified, superseded, or canceled, at any time by TPS. The Agreement shall be governed by and construed and enforced in accordance with the laws of the state of California, without regard to conflicts of laws provisions. Any action concerning this Agreement shall take place in the state or federal courts located in San Francisco, California. If for any reason any provision of this Agreement, or a portion thereof, shall be unenforceable, that provision shall be enforced to the maximum extent permissible so as to effect the intent of this Agreement, and the remainder of this Agreement shall continue in full force and effect. This Agreement constitutes the entire agreement between us and you with respect to the Software and it supersedes all prior or contemporaneous communications, agreements and understandings between TPS and you with respect to the subject matter hereof. No joint venture, partnership, employment, or agency relationship exists between you and TPS as a result of this agreement or use of the Service. The failure of TPS to enforce any right or provision in this Agreement shall not constitute a waiver of such right or provision unless acknowledged and agreed to by TPS in writing. A printed version of this Agreement shall be admissible in judicial or administrative proceedings.

BY CLICKING ON THE “ACCEPT” BUTTON BELOW YOU ACKNOWLEDGE THAT YOU HAVE READ AND UNDERSTOOD THIS AGREEMENT AND AGREE TO BE BOUND BY ITS TERMS. YOU FURTHER AGREE THAT THIS AGREEMENT SUPERSEDES ANY PREVIOUS AGREEMENT, WHETHER WRITTEN OR ORAL, RELATING TO THE SOFTWARE BETWEEN YOU AND TPS.

Table of Contents

Introduction

MAGNET Field Site is a cloud enabled data collection software for construction measurement and layout activities and includes data collection, stakeout functionality and computations, and connects to Topcon Robotic and RTK systems.

It is a part of the MAGNET family of products that includes MAGNET Office and MAGNET Enterprise.

After successfully installing the application on your device, the Product Activation wizard guides you through the activation process.

Note:

- After installing an update to MAGNET Field Site, the application will need to be activated again.
- If the activation is canceled, MAGNET Field Site will run in Demo mode. Demo mode, allows user to store up to twenty five points and create a road of one hundred meters in length.
- MAGNET Site can be activated at the time of installation directly from your PC.

To establish a connection with MAGNET Enterprise and exchange data, follow the connection prompts after activating the software.

Note: MAGNET Enterprise connection is not available for MAGNET Field Site Onboard.

Once a connection with a device is established, the Home Screen is displayed. Alternatively, the user can click the Home Button to open the Home Screen for the default job.

At this point, MAGNET Field Site has been successfully configured and a new job can be created. The user-friendly interface will provide intuitive settings and a simple approach to managing tasks.

On each subsequent MAGNET Field Site startup, the connection dialog runs with the current job selections.

Home screen

See the Introduction for general information on MAGNET Field Site.

Click an icon to open the folder or execute the command:

Job
Manages jobs.

Map
Shows the current job data on the map.

Setup
Sets up surveying: GPS+ survey or Optical (Robotic) survey.

Topo
Conducts a survey with static points and determine inaccessible points by setting offsets from measured points.
Find out more on Offsets.

Surface Topo
Creates or appends to a real-time Surface surface.

Auto Topo
Conducts a survey with dynamic points.

Stake Point GPS
Stake Point Optical (Robotic)
Stakes design points.

Stake Polyline
Stakes points on a linework selected either by CodeString, from a linework list, or from the map.

Stake Road
Stakes points on a road.

Stake Slope
Stakes points near a slope on the desired alignment.

[Surface Check](#)

There are three types of Stake Surface: Elevation, Road, and Surface.

[Volumes](#)

Calculates the cut, fill and area between two surfaces.

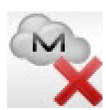
The top banner of the home screen shows the name of the job that is open and associate icons. More...

Home screen's associate icons

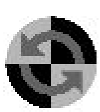
Options

Provides access to Help files and options that are specific to the current open screen. Over this icon in the Home screen and in every dialog for measurements, a blinking notification displays if you received a file  or a message . The file is available in the 3DMC Inbox folder and the message in Chats.

Indicates the power status of the controller battery.

Indicates the successful connection status with the Enterprise. When connecting, the animated icon  displays. A red cross  shows that the connection is off. You can connect/reconnect with the enterprise in the Connections dialog.

Indicates the successful connection status with the SiteLINK 3D server.



Indicates the connection status with the device. You can connect/reconnect with the device in the Connections dialog. If the device is disconnected, the indicator is grey. When connecting, it turns yellow, then green, and finally turns into the image of the connected device with a green check mark near it.

Closes the program.

Returns to the home screen.

Help Options

Click the icon to open a menu of options which may be useful for you. Every screen and dialog in MAGNET Field Site has such an icon in the top left corner, the menu, opened by clicking the icon, always contains the Help option and may contain some additional options specific to the current screen.

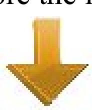
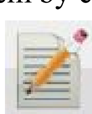
The following options can be selected from the menu of the home screen and every folder:

- Click *Help* to access help files.
- Check mark *Keyboard* to enable you to type entries in a dialog's fields using the soft keyboard.
- Click *Chats* -> Create New to configure a chat with the MAGNET Enterprise users. Find out more...
- Click *Timecard* to open the *Timecard* dialog when the Enterprise connection is established. Find out more...
- Click *Configure Menus* to show / hide functions in the menus. Find out more...
- Click *Activate Modules* to perform a license activation with the help of the Activation Manager wizard.
- Click *Minimize* to reduce the current window to a button on the taskbar.
- Click *Software Updates* to check whether any updates are available for MAGNET Field Site. Find out more...
- Click *About* to view basic information about MAGNET Field Site software. Find out more...

Configure Menus

This dialog enables you to change the contents of the Home screen or the folders within it. Bear in mind that maximum TWELVE menu items can be shown on the screen. If there are more than twelve items, only first twelve of them will be shown.

To configure menus:

1. Highlight the name of the desired Home screen's item. The other panel will show the contents of the corresponding folder. By default MAGNET Field Site displays full contents of all the folders.
2. If you want to hide a seldom or never used item in this folder, clear the box near the desired name. Select the box again to restore the item in the folder.
3. Use  and  icons to move the highlighted name up and down in the list to change the order of icons.
4. You can cut a selected item by clicking  and paste it above another selected item with .
5. To rename an item, click  and enter a new name.

6. Click  to create your password to freeze menus for editing if required.
7. Click  to store the changes and open the modified Home screen.

Software Updates

This dialog shows available updates detected of MAGNET Field Site. The dialog consists of two parts - upper part shows general update info and info about available features updates. The other part shows info about available program version for update, or, if none, current version info.

- Click the **Check** button to perform check for all updates (features and program versions), renews info in dialog in case of success.
- Click the **Apply** button to start the features updating.
- Click the **Details** button to show the release notes for available program version.
- Click the **Install** button to install a new version of MAGNET Field Site.

About MAGNET Field Site

This lets you:

- View information about the current version of MAGNET Field Site.
- Get the Id of the current device.
- Learn the privacy policy of the company. For this, click the **Privacy Policy** button.
- Collect support information for the support team. For this, click the **Support Info** button. More...

Collect Support Information

This dialog lets you collect all information needed for support into an archive file:

1. In the **Support Info filename** field enter the name of the target archive file. By default, it will be the name of the current job.
2. In the comment area, you can add any comments which may help the support team resolve your issue. This information will be saved in the target archive file as a text file.
3. If required, select the **Upload support info via Internet** check box to transfer the archive file to the Enterprise server for the support team. This is available only when you are registered for Enterprise.
4. Click  to save the archive file in the current job folder by default and optionally to send it to Enterprise.

Port Logging

Displays traffic with the currently connected device. Allows you to enable file logging.

The icon  opens a pop-up menu that contains the following options:

- Check mark *Show incoming port data* to view incoming traffic information of a controller port connected to MAGNET Field Site.
- Check mark *Show outgoing port data* to view outgoing traffic information of a controller port connected to MAGNET Field Site.
- Check mark *Pause logging to screen* to freeze port traffic show.
- Select the *Log to file* check box to save the data to a file. The Save File dialog displays. Assign a name and select a desired directory in which to save the file on the controller. By default, the *comm.log.txt* file will be saved in the MAGNET Field Site folder. Pressing the ok button returns to the Port Logging screen to view the data being saved to file. The file name is also shown on the screen
- Select the *Append* check box to add new data to the existing file. Click the ok button to confirm the operation.

Job Folder

Jobs include all data collected in surveys of different configurations. Upon installation, MAGNET Field Site only contains and opens the default job.

Click an icon to manage your jobs:

New Job
Creates a new job.

Open Job
Makes a selected job current.

Delete Job
Deletes a selected job from data storage.

Job Info
Shows information about a current job.

Save Job As
Copies a current job with a new name.

Configure
Manages job configurations.

Exchange
Exchanges data between an active job and other jobs, files and enterprise projects.

Edit
Edits existing job data.

Calculate
Calculates various coordinate geometry tasks.

Chat
Chats with online recipients.

News
Allows you to view news and other information from Magnet RSS.

Apps
Uses applications that are pre-installed on the same device as the MAGNET Field Site.

New Job

The new job creation process is performed with the help of a Wizard.

indicates the directory in which this job will be created. The last specified file path is retained.

To create a new job:

1. Enter a **Name** for the job.

   Note: The Job name length should not exceed 63 characters and contain any of the following symbols:

   ! , ? % * @ # $ % ^ & ' " \ / | ~ ; [ ] { } ( ) < > ` + = .

2. Optionally, in **Created by**, enter any identifier of the person creating the job.
3. Optionally, in **Comments**, enter a description for the job.
4. ***Current date*** displays the date and time when the job is being created.
5. Click Browse if required to save the New Job file in a folder different from that indicates.
6. Click at this step if you want the new job to use the settings from the last opened job as the default settings. The new job becomes the current job, and the Home menu displays with the job name in the title area.
7. Click **Next** to follow the wizard to complete the required steps.

   Note: Clicking during any of the following step creates the new job with the settings that were configured in all the previous steps (including the opened one).

8. Select the **Survey Style** or create a new configuration, and click **Next**. A Survey Configuration is a set of parameters that does not depend upon the job. A Configuration can be used by different Jobs. More...
9. Configure the **Coordinate System** settings required for the job, and then click **Next**. More...
10. Specify the **Units** for the job, and then click **Next**. More...
11. Configure the **Display** parameters for coordinates, azimuths and positions on roads, and then click **Next**. More...
12. Set **Alarms** for various situations. More...

13. Click  to open the Home menu for the newly created job. The title of the Home menu will be the current job name. When opening the job the *Connections* screen displays by default.

Open Job

Upon starting, MAGNET Field Site always opens the last used job after the product activation and Connections are performed.

All existing jobs that are created/opened with MAGNET Field Site are defined as **.mjf** files and have the symbol .

To open an existing job:

1. From the **Job Name** list, highlight the name of the job you want to open.
   The **Created** and **Modified** fields will display when the job was created and last modified.
   will show the directory in which this job resides. By default the job files are stored in the [program]\Jobs folder.
   By default the list displays the jobs in the order of last opening. If required, click the Job Name header to sort the jobs in the alphabetical order, click again to return to the order by date.
2. Click Browse to navigate to the desired job in another folder if required.
3. Click to open the job. The Home menu displays.

To open job backups of existing jobs:

1. Click Browse to navigate to the desired job.
2. In the File Types drop-down list, select MAGNET Field Site Job Backups (*.mjf.bak).
3. Highlight the name of the job to be opened, and click .

To open a job that was created in TopSURV:

1. Click Browse to navigate to the desired job.
2. In the File Types drop-down list, select TopSURV Job Files (*.tsj) or TopSURV Job Backups (*.tsj.bak).
3. Highlight the name of the job to be opened, and click .

 Delete Job

To delete a job:

1. From the **Job Name** list, highlight the name of the job you want to delete.
   The **Created** and **Modified** fields will display when the job was created and last modified.
    will show the directory in which this job resides. By default the job files are stored in the [program]\Jobs folder.
2. Click Browse to navigate to the desired job in another folder if required.
3. Click  to delete the job. A confirmation message displays.
4. Click **Yes** to confirm deletion or **No** to cancel, and return to the Job folder.

Note: When deleting a job, MAGNET Field Site automatically deletes all associated files (the job history file, images, etc.).

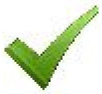
Job Information

Displays information about:

- The current job in general
- The settings in the job
- The currently connected receiver:
  - Receiver firmware version
  - Receiver OAF expiration date (for nearest expired option). Click on this date to expand a full list of OAF options.

Save Job As

To save the currently opened MAGNET Field Site Job file with a new name:

1.  displays the path to the current folder. Use the dialog icons to navigate to the desired folder to save the new file. Learn more about the icons from Browse.
2. Enter the **Name** of the new file.
3. Click  to perform the operation.

Browse

The title of this dialog depends on the path from which it is called.

 displays the path to the current folder where you will search for a desired file:

1. Use an icon to:

    move up a folder

    create a new folder

    switch the display mode for the list of files to the detailed view

    switch the display mode for the list of files to the icon view

2. Select the **Type** of files from the drop-down list to display the files in the list for the current folder.
3. Enter the **Name** of the file to select or select the desired file from the list.

Configure folder

Click an icon to configure the settings:

Equipment
Configures equipment to work.

Coordinate System
Sets a coordinate system for the job. Allows Grid to Ground transformation (find out more...).

Global
Logs the current job history to the file, and connects with the instrument at the startup prompt.

Backup
Changes the directory to save job backups.

Units
Sets default units to be used in the job.

Temp/Press
Sets the temperature and air pressure for total stations.

Display
Customizes the interface to display data in the job.

Alarms
Sets alarm parameters.

Codes
Sets code global parameters.

Stake Reports
Configures reports for staking.

Cloud Connections
Configures the user account to enter into the MAGNET Enterprise.

Select Survey Configuration

Survey Configuration is a set of parameters that control a survey, define communication between devices, measure and store points. MAGNET Field Site contains some default configurations for different types of surveys with GNSS receivers and optical instruments. If the defaults do not suit your needs, you can create a new configuration for your survey. To do this, click . When you have created the new configuration, the name of this configuration will appear in the list.

To set a survey configuration for the job:

- From the **GPS+ Configuration** drop-down list of available configurations, select the required configuration.
- For the **Optical Configuration** only the My Robotic configuration is available.
- Both types of configurations can be selected for one job. They will be used with the device selected upon Connections: GPS+ or Optical.
- Hybrid Positioning is available for selection when using any GPS+ continuous and only Robotic optical configurations for one job.
  Hybrid Positioning mode implies simultaneous connection with both GPS+ receiver and optical robot. In this mode you needn't connect/disconnect with the current device.
- Automatic Localization is available when **Hybrid Positioning mode** is activated. First **five** points will be used for calculating parameters between WGS-84 and Local coordinate systems. These parameters are saved in the job and they can be automatically updated during next measurements in the Hybrid Positioning (if residuals for these points will be less than for the previous set of points).
- After selecting the survey configurations for your job, click to store settings, which will be used each time you start a survey, and to return to the Home screen.

Note: Both types of configurations can be selected for one job. They will be used with the device selected upon Connections: Optical or GPS+.

Configurations

The Configurations dialog presents a list of available survey configurations: names and types. Survey configurations are stored in the *Styles.tsstyles* file in the MAGNET Field Site directory. MAGNET Field Site contains a list of predefined configurations that you can use or modify.

You can customize the list of the configurations to include only ones you need:

- Highlight the name of the configuration.
- Click the **Delete** button to remove the configuration from the list.
- Click the **Edit** button to change the highlighted configuration to suit your survey preferences.
- Click the **Add** button to create a new configuration.

Creating/editing a survey configuration is accomplished with the help of a Wizard. Find out more... on GPS+ configurations and more... on the optical configuration.

Coordinate System

To set the coordinate system for the Job:

- Select the **Projection** to be used in the job. MAGNET Field Site has many pre-defined projections and tabular projections, which are selected during the MAGNET Field Site installation.
- *Use* **Grid/Ground** to covert to ground coordinates if required.
- Specify the **Datum** as required. MAGNET Field Site has a lot of cataloged datums.
- Select the **Geoid** to bring to elevations if necessary. MAGNET Field Site has a lot of geoid files, which are selected on MAGNET Field Site installation.

Find out more...

Units

You can set different units and precisions for how MAGNET Field Site will display various numerical values in dialogs:

- Distance (including area and volume)
- Angle
- Coordinate precision
- Temperature and pressure

On the Distance tab, select as required:

- **Distance unit** for any length values. That can be Meters; IFeet - (1 International ft = 0.3048 m); US Feet (1 US survey ft = 1200/3937 m); IFeet and Inches, US Feet and Inches (the latter two are calculated by taking into account that 1 ft = 12 in), IChains (1 International ch = 66 International ft) or USChains (1 US survey ch = 66 US survey ft).
  Note: If the selected units are USfeet, linear values can be entered as meters or IFeet by appending "m" or

"if" to the entered value. If the selected units are in meters, then a linear value in USFeet or International feet can be entered by appending "f" or "if" to the end of the entered value. If the selected units are in IFeet, linear values can be entered as meters or USfeet by appending "m" or "f" to the entered value. The appended characters "m", "f", or "if" are case insensitive. In other words, enter "M", "F", or "IF".
Note: When using IFeet and Inches or US Feet and Inches the following format is observed: f.iix, where f is feet, ii is inches and x is 1/8th of an inch.

- **Distance precision** for the number of decimal places in length values.

  Setting precisions
  To display only integers, select "0" and so on. To display 5 decimal places, select "0.12345".
- **Area unit** for area values
- **Volume unit** for volume values

On the Angle tab, select as required:

- **Angle unit** for angular values. That can be *DMS*, represented as ddd mm ss (the full circle contains 360 degrees) or *Grads (Gons)* - the full circle contains 400 grads (gons).
  Note: Azimuth can be entered as two points separated by "-", "," or ";". Certain angles can be entered as three points separated by "-", "," or ";". For instance a value of 100-101 indicates the Azimuth from Point 100 to Point 101.
- **Angle precision** for the number of decimal places in angular values. More...
- **COGO angle unit** for angular values in COGO tasks. In addition to *Angle unit* settings, that also can be *Radians* (the full circle contains 2*PI radians); or *Mils* (1 Mil = 1 Milliradian = 1/1000 of a Radian)
- **COGO angle precision** for the number of decimal places in angular values in COGO tasks. More...

On the Coordinate tab, select as required:

- **Northing/Easting precision** for the number of decimal places in Northing/Easting coordinates. More...
- **Lat/Lon precision** for the number of decimal places in latitude/longitude seconds.
- **Height precision** for the number of decimal places in ellipsoidal heights and elevations. More...

On the Other tab, select if necessary:

- **Temperature unit** only for the raw measurements. That can be Celsius (C) or Fahrenheit (F).
- **Pressure unit** only for the raw measurements. That can be mmHg, hPa, inHg or mbar.

Display

You can configure the display of the following settings:

- **Coordinate Type** to view coordinates for the coordinate system selected
- **Coordinate Order** to display: Northing/Easting/Height or Easting/Northing/Height, X/Y/Z or Y/X/Z
- **Azimuth Origin** to display the azimuth computed from either reference directions: North, South, East, or West
- **Disp Dir As** to view directions as bearings or azimuths
- **Disp CL Pos As** to display stations or chainages (distance along the centerline) for stationing the centerline. For example, when using 100 m as a station, a position at the distance of 341.256 m (chainage) from station 0 + 00 is designated 3 + 41.256. The numbers to the left of the plus are hundreds of meters (even full stations) and to the right of the plus are tens and units of meters to the nearest 0.001 m (a remainder of a station).

Alarms

You can configure alert conditions:

- On the Main tab, select **Audible Alarm** to enable audible alarms. The alert will sound automatically when an alert situation occurs. For GPS: select **Enterprise Alarm** to enable a sound when new chat comes; select **RTCM.3.x Coordinate Systems** to enable a warning that the coordinate system is set by reading the RTCM messages.
- On the Controller tab, place check marks in the **Power Alarm** and **Memory Alarm** boxes to select the alert conditions of low power and memory in the controller.
- On the GPS+ tab, place check marks in the **Power Alarm** and **Memory Alarm**, **Radio Link** and **Fix-Float** boxes to select the alert conditions of low power and memory, poor radio link and loss of initialization in the GNSS receiver.
- On the Optical tab, place check marks in the **Power Alarm** and **Track** boxes to select the alert conditions of low power and loss of the target in automatic tracking mode for optical instruments. Select **Grid/Ground Warning** to enable a warning when a grid coordinate system is set.
- Click  to observe information about the controller and the connected device status.

GPS Survey

To create a new configuration:

1. Enter the **Name** for the configuration that will be displayed in the Configurations list. MAGNET Field Site includes a number of predefined configurations you can select and edit for a new configuration.
2. Select the survey **Type** that depends on the equipment you will use and the results you need to obtain. Every survey type has a predefined configuration with the name of My followed by the survey type (e.g. My RTK). Network RTK survey type may have the additional special configuration My MAGNET Relay. My MAGNET Relay configuration is created only on an empty controller.

   MAGNET Field Site provides configurations for the following GPS+ survey types:

   RTK

   Network RTK

   Real Time DGPS/NMEA
3. From the **Corrections** drop-down list, select the type of correction data that will be used for survey if the Network RTK survey type is chosen.
4. Click **Next**. The wizard will set default parameters specific for the equipment selected.

Receiver Make

Set general conditions for the survey and click **Next**:

- To access GPS real-time surveys and stake routines without using real hardware, select the **Simulation mode** check box.
- In the **Rover** field, designate the **Manufacturer** of the rover receiver, MAGNET Field Site supports Topcon and Sokkia.
- In the **Base** field, designate the **Manufacturer** of the base receiver, if required.

RTK Survey

Real-Time Kinematic survey implies use of a pair of receivers operating simultaneously and a radio link established between the two receivers. From a functional point of view, the two receivers will differ from each other. One of the receivers (a Base) is located at a fixed point with known coordinates. The base receiver will transmit the data via a radio link to the other receiver (a Rover) to compute differential corrections. To establish a proper connection between the two receivers, specify the necessary communication parameters first.

To configure an RTK survey:

- Configure the base receiver
- Configure the base radio
- Configure the rover receiver
- Configure the rover radio

Base Receiver

To configure the base receiver:

1. The **Ext. Receiver** box is enabled for a controller that has an internal GNSS receiver. You can select a connection with the internal GNSS receiver or any external GNSS receiver. If the controller does not have a GNSS receiver or the software is installed on the computer, the **Ext. Receiver** box is always checked. The connection for external receivers can be *Bluetooth* and *Serial*.
2. From the **Receiver Model** list, select the model of the receiver you are using as the base. *HiPer SR* is the default.
   Note: *Topcon Generic* selection will work with all receiver models including GR-3 but some functionality, like automatic SBAS satellites tracking, will not be accessible.
3. Optionally, enter the **Serial Number** of the receiver.
4. Set **Elevation Mask**. Usually the default 13 degrees is appropriate. Data from satellites below this elevation will not be used.
5. From **RTK Format**, select the format for the differential corrections data transmitted by the base radio.
   For RTK configuration, the RTCM MSM format will be supported in Topcon GNSS receivers with firmware 4.5 and later .
   For MAGNET Relay configuration, the Relay Name field will show the user's MAGNET Field Site license serial number as the default relay name on the given controller. This name is used when the Enterprise server generates a mount point name and the Start Base dialog. If there is no serial number yet, the Relay name is offered.
6. From the **Antenna** drop-down list, select the type of the antenna used at the base.

7. Enter the antenna height and set the type of height measurement (vertical or slant). To do this, click the **Antenna Height** button and select *Edit* from the drop-down menu. The Antenna Setup dialog is opened. Here you can edit the antenna type, the value of the antenna height, and the type of height.the **Antenna Height** and height type.
8. Click **Peripherals** to enable the Multiple Ports option if needed to transmit data from different ports of the base receiver to peripherals.
9. Optionally, you can turn off the charger mode for the receiver's internal battery if it is available. To do this, select the Receiver Settings option from the pop-up menu that displays by clicking in the top left corner. More...
10. Click **Next**.

Base Peripherals

Select the **Multiple Ports** check box, and then select the number of ports to use for base output if required.

Additional dialogs will appear later in the wizard sequence to setup parameters for each port.

Receiver Settings

When an external power supply is used at the base, and the receiver charger mode is turned on, it supplies power to the receiver battery. You can use an external power supply without charging the receiver battery.

To do this:

1. Select the **Turn Charger Mode Off** check box.
2. Click to save the setting.

Radio

Radio modems are used in GPS real-time surveys. The base radio is intended for transmitting differential corrections and the rover radio is used for receiving them. Also, there can be some radios that may be configured for output NMEA messages if required.

To configure the radio:

1. Select the **Radio Modem** type from the list of pre-defined modem types. The list contents depends upon the survey configuration and the receiver model selected. For HiPer SR receivers, the Long LINK modem does

not require any further settings.

Note: For a Network survey, in the *Network Type* field, select either existing network connection or dialup.

2. If you use a AirLink CDMA or GPRS radio modem, select the *Modem Register* option from the pop-up menu that displays by clicking  in the top left corner to register the radio modem. More...
3. Optionally, you can configure RE-S1 Repeater for the rover radio. To do this, select the *Config RE-S1 Repeater* option from the pop-up menu that displays by click  in the top left corner. More...
4. Specify communication parameters for the receiver Port that the radio is connected to: **Parity**, the number of **Data** bits, **Baud** rate, the number of **Stop** bits, which are specific to the connected modem.
5. If required, click **Defaults** to return all the communication parameters to the default settings.
6. Click **Next** to specify the Radio Parameters.

Modem Register

AirLink GPRS and AirLink CDMA require registration.

To register the modem:

1. Select **Provider**.
2. Enter *Register Information*: **Net ID**, **Password** and **Phone Number**.
3. Click the **Defaults** button to set the default data.
4. Click the **Register** button to pass registration.

Radio Parameters

Specify parameters for the radio selected:

- AirLink GPRS, CDMA, CDPD1, CDMA2000, Generic, Sierra Wireless MP200 CDPD modem types do not require setting any parameters for the base radio.
- Other modems require additional parameters to be set.

For more information see:

AirLink CDMA(MUDP)

Digital UHF

FH915Plus

Internal Satel

Internal HiPer Lite Radio

[Pacific Crest]

[UHF]

[Satel]

[Cellular Modem]

[Internal CDMA]

[Rover AirLink CDMA]

[Rover AirLink CDPD]

[Rover AirLink GPRS]

[Rover Internal CDMA]

Base Multicast

To set IP addresses for communication between the base and several rovers using UDP protocol:

1. In the **Address to Add** field, enter the IP address as required.
2. Click **Add** to add the IP address to the list.
   The *IP Addresses list* displays all IP addresses available.
3. Click **Delete** to delete any address from the list.

Cell Phone

To set phone information:

1. For the base radio, enter the **Base PIN**.
2. For the rover radio, enter the **Rover PIN** and the **Base Phone Number** that can be added to a *Phone number list.*

Digital UHF

To set parameters for the Digital UHF radio:

1. Select the **Revision** type of digital UHF radio used: either *Digital UHF* or *Digital UHF II*.
2. Select an appropriate operation **Protocol** for data transmitting/receiving:

   - *Simplex* is the default. This is ArWest's proprietary protocol. It only works with other ArWest (Digital UHF) radios.

   - *Trimble* works with Trimble Trim Talk and Trim Mark Radios.

- *PDL* works with existing PDL radios and Hiper XTs set in PDL mode.

3. Select the type of **Modulation** for the radio modem. Select either DBPSK/DQPSK if using the Simplex protocol or GMSK if using Trimble or PDL.
4. Select the channel **Spacing** step: either 12.5 or 25 kHz or Do not set (the default). This is a hardware set.
5. For *Digital UHF II* and *PDL* protocol, in **Scrambling** set either Don't set up (the default), Off or one of the values to provide more robust data communication over high interference areas.

FH915 Plus

To set parameters for the FH915 Plus radio:

1. Select the **Location** to adjust the frequency range and RF power level depending on the country.
2. Set the operation **Protocol**. Select *FH915 Ext* if all receivers on the jobsite are equipped with FH915Plus radios.
3. Set the **Channel** number. This feature allows up to ten simultaneously transmitting radio modems, without interference, at the job site (1 is the default).

Int Satel Parms

To set parameters for the internal Satel radio:

1. Select **Satel FCS On** (default), or **Satel FCS Off**. In Free Channel Scan mode, the radio modem will regularly scan the current operating frequency and under certain conditions it can switch to the next frequency.
2. Or select **PDL** (PacCrest) operation protocol.

Internal HiPer Lite

Set the **Channel** number. This feature allows up to five simultaneously transmitting radio modems, without interference, at the job site (1 is the default)

Pacific Crest Radio Parms

To set parameters for the Pacific Crest radio:

1. Select the **Channel** number from 0 to 15.
2. Set the **Sensitivity** of the radio to *Low*, *Medium* or *High,* or turn it *Off.*

UHF Modem

To set parameters for the internal UHF radio:

1. Select the compatibility **Protocol**.
2. Set the **Channel** number.
3. Select the output **Power** value for the base radio.

Modem Dialup Info

To set parameters for a dialup Internet connection:

1. Enter the **Dialup Number** needed to make the Internet connection.
2. Enter the **User ID** for the server.
3. Enter the **Password** to login to the server with the entered user id.
4. Enter the **PIN** number for the server.
5. To set the values to default, click the **Defaults** button.

Satel Radio Parms

To set parameters for the Satel radio:

1. Select the **Model** of the Satel modem.
2. Select the **Channel** number. **Frequency** displays the corresponding value for the channel number selected.
3. If required, set **PCC Mode** (Pacific Crest Corporation).
4. In **FEC**, select if you want to use Forward Error Correction technique to maximize data communication.

Rover Receiver

To configure the rover receiver:

1. From the **Receiver Model** list, select the model of the receiver you are using as the rover. *HiPer SR* is the default. Note: *Topcon Generic* selection will work with all receiver models including GR-3 but some functionality, like automatic SBAS satellites tracking, will not be accessible.
2. The **Ext. Receiver** box is always checked and unavailable for every receiver model in the Receiver Model list except GRS-1. MAGNET Field Site directly installed on the GRS-1 can use GRS-1's internal GPS receiver.
3. Optionally, enter the **Serial Number** of the receiver.
4. Set an **Elevation Mask**. Typically the default 13 degrees is appropriate for the base and rover. Satellites below this elevation will not be considered.
5. From **RTK Format**, select the format for the differential corrections data received by the rover radio.
6. In Network survey, set the **Protocol** for data transmission: *NTRIP*, direct *TCP/IP*, or *CSD* technology. MAGNET Relay is set by default for the protocol in the MAGNET Relay configuration.

7. From the **Antenna** drop-down list, select the correct antenna type used on the rover side.
8. Optionally, enter the **Serial Number** of the antenna.
9. To set the default antenna height, enter a value in the **Antenna Height** field. The height type can be set to either *Vertical* when the height is measured to an ARP (the antenna reference point, usually the middle bottom of the antenna) or *Slant* when the height is measured to the antenna edge.
10. Click **Peripherals** to enable options to configure selected peripherals in the dialogs later. More...
11. Optionally, you can turn off the charger mode for the receiver's internal battery. To do this, select the *Receiver Settings* option from the pop-up menu that displays by clicking in the top left corner. More...
12. Optionally, you can instruct the rover to use relative calibrations on the base. To do this, click *Relative calibrations on base* in the pop-up menu (the check will appear near the option name). The default is the use of absolute calibrations. All antenna calibrations are referenced to the ARP. You can find details on Antenna Calibrations at the NGS site.
13. In Network RTK survey you can set a fixed position for GGA message from the rover receiver. To do this, click *Use fixed GGA position* in the pop-up menu to enable and *Set fixed GGA position*. More...

Rover Receiver Peripherals

If required, select any options to work with peripherals:

1. Select the **NMEA Ports** check box to configure the output of the NMEA messages. Select the number of ports from the drop-down list that appears and click . Additional dialogs will appear later in the wizard sequence to setup parameters for each port and specify NMEA messages for output. More...
2. Select the **Multiple Ports** check box to configure multibase input for the rover. Select the number of ports from the drop-down list that appears and click . Additional dialogs will appear later in the wizard sequence to setup parameters for each port. Note: Use only one radio to receive corrections from the base.
3. Select the **Depth Sounder** check box to configure a depth sounder. Click **Parameters** to specify settings. More...
4. Select the **mmGPS+** check box to configure the mmGPS+ system. Click **Parameters** to specify settings. More...
5. Select the **External Laser** check box to configure the laser device that can be connected either directly to the *Controller* or through the *Receiver*. Click **Parameters** to specify settings. More...

Position for GGA

If required, set a fixed position for GGA message from the rover receiver.

You can do this in one of the following ways:

1. Click the *Map* selection button and select the point on the map. The point position values appear in the coordinate fields.
2. Click the *List* selection button and select the point in the list. The point position values appear in the coordinate fields.
3. Enter the coordinate values in the edit fields.

Rover CDMA Parms

Set IP address for communication between the rover and the base.

- Enter the CDMA address of the base radio in the **Base CDMA Address** field.
- Press **Add** to add the address to the *IP Addresses list*.

Rover CDPD Parms

To set the IP address for communication between the rover and the base.

1. Enter the CDPD address of the base radio in the **Base CDPD Address** field.
2. Press **Add** to add the address to the *IP Addresses list*.

Rover GPRS Parms

To set the IP address for communication between the rover and the base.

1. Enter the GPRS address of the base in the **Base GPRS Address** field.
2. Press **Add** to add the address to the *IP addresses list*.

Modem Internet Info

To enter information for Internet connection:

1. Enter an Internet **Address** that will be used for the connection in the *IP* or *Web* format (select the radio button as required).
2. If required, enter a **Name** for the address which displays in the address list.

3. Press **Add New** to add the address to the *Address list.* New IP/Web addresses/ports can be deleted or added to the list.
4. Press **Update** to update the address in the address list.
5. You can select an address by double-clicking on it in the list.

RE-S1 FH915 Repeater

If required, the RE-S1 modem can be used as a stand-alone repeater to increase the range between the Base and Rover in spread spectrum systems.

To configure the RE-S1 Repeater:

1. Select the **Enable RE-S1 Repeater configuration** check box to enable further configuration of the RE-S1 as a repeater.
2. From **Connect Type**, select the type of connection of the repeater: either to the *Receiver* or to the *Controller* to setup the modem.
3. In **Connect Port Setup** set the properties of the connection port. Press *Defaults* to set the port properties to the default values.
4. Press **Next** and configure the modem parameters:
   - Select the **Location** to adjust the frequency range and RF power level depending on the country.
   - Set the operation **Protocol**.
   - Set the **Channel** number. This feature allows up to ten simultaneously transmitting radio modems, without interference, at the job site (1 is the default).

Output NMEA

To configure the *NMEA Messages* for output:

1. Select the check boxes near the types of messages as required. More...
2. Select the **Set GP as Receiver Talker ID** check box to instruct the receiver to use "GP" as Talker ID in appropriate NMEA sentences generated. This enables support of Goggle Maps that cannot recognize default "GN" or "GL" as Talker IDs in these messages.
3. Enter the **Interval** in seconds in which the application will output the messages (up to 0.1 sec.).

List of NMEA messages

The following *NMEA Messages* are available for output:

- *GSA* to output the operation mode of the GNSS receiver, the satellite used for positioning, and DOP.
- *GLL* to output data on the current latitude/longitude and positioning mode.

- *VTG* to output the traveling direction and velocity.
- *GRS* to output the residual error of distance for each satellite. This is used to support RAIM.
- *ZDA* to output the UTC, day, month, year, and local time zone.
- *GST* to output the statistics of position errors.
- *GNS* to output data on time, position, and positioning of GPS+GLONASS (GNSS).
- *GGA* to output data on time, position, and positioning.
- *GSV* to output the number of satellites, satellite number, elevation angle, azimuthal angle, and SNR.
- *HDT* to output the direction (heading).
- *P_ATT* to output attitude parameters.
- *RMC* to output time, date, position, course and speed data provided by a GNSS navigation receiver.
- *ROT* to output rate of turn.
- *GMP* to output GNSS map projection fix data.

Config Depth Sounder

If required, select **Simulation Mode** to test and demo depth sounder functionality without actually having a depth sounder in water.

To configure the *Depth Sounder*:

1. Select the **Model** of the depth sounder. Currently only HydroLite-TM is available
2. Set **Depth Sounder Port Setting** including port, parity, data, baud and stop rates to connect to the device.

When the configuration is used, the depth sounder icon will appear on the status bar of any measurement dialog.

---

mmGPS+ Settings

In mmGPS aided RTK survey, a wireless sensor connected to the Rover picks up the signals from the laser transmitter for accurate (millimeter) elevations.

Note: When measuring the height of the Rover antenna, include the height of the sensor with a 5/8 inch plug.

To configure the mmGPS+:

1. Select the **Receiver port***,* which is connected to the mmGPS+ Sensor.
2. Select the **Sensor Gain** to adjust the gain on the mmGPS+ Sensor. Select Auto to automatically control the mmGPS receiver's detection level of the transmitter's signal.
3. Enter the **Height Difference Limit** value to set the threshold for the difference between GPS and mmGPS+ height measurements. If the GPS+ height and mmGPS+ height differ by more than the amount entered, the mmGPS+ icon will change to warn the user.

Laser Config

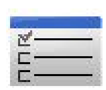
To configure the *External Laser* instrument connected to the *Receiver* or to the *Controller*:

1. Select the laser **Manufacturer**. Currently MAGNET Field Site supports MDL and Laser Technology, Inc.
2. Select the **Model** of the Laser Instrument.
3. Select the **Type** of laser measurement system if it uses Encoding or not.
4. Set **Laser Port Setting** including port, parity, data, baud and stop rates to connect to the Laser Instrument.

Topo Survey Settings

To configure parameters that will be used during a real-time stationary survey:

1. In the **Precise** area, set the parameters for storing positions:
   - From the **Solution Type** drop-down list, select the solution type for each position computation as required. Data will be considered only if the solution type satisfies this selection. What is displayed in the list for selection depends on the type of selected survey. More...
   - Select the **Meas Continuously** check box to log measurements continuously and stop them manually when required.
   - The **Average** field allows you to set a specific number of measurements for logging and average them for storing the position. If required, you can change the default number (3) of measurements.
   - Select the **Precision** check box to consider the threshold of the horizontal and vertical precisions of measurement to store the position. The defaults are 0.015 m and 0.030 m, respectively.
     Note. When the **Precision** condition is satisfied and:
     Meas Continuously is enabled, the logging immediately stops to store the point.
     Meas Continuously is disabled, the **Average** position is calculated to store the point.
   - Select the **Auto Store** check box to enable auto-storing positions for the average mode.
2. In the **Quick** area, set the parameters for fast auto-storing positions:
   - From the **Solution Type** drop-down list, select the solution type for each position computation as required. More...
   - In the **Average** field, if needed, you can change the default number (1) of measurements required for storing positions.
   - Select the **Precision** check box to consider the threshold of the horizontal and vertical precisions of measurement for storing positions. The defaults are 0.015 m and 0.030 m, respectively.

Note: Every survey parameter can be changed with the help of the  button from any Survey screen in GPS+ mode.

Solution Types

There can be a combination of the following solution types:

- *Fixed mmGPS+*: positions were Fixed Only solution with mmGPS+ calculated height.
- *Fixed Only*: positions were computed by RTK engine using the carrier phase measurements from Base and Rover receivers. Integer ambiguities were fixed.
- *Float mmGPS+*: positions were Float solution with mmGPS+ calculated height.
- *Float*: positions were computed by RTK engine using the carrier phase measurements from base and rover receivers. Integer ambiguities, however, were NOT fixed (their float estimates were used instead).
- *DGPS*: positions were achieved using the pseudo-range measurements from base and rover receivers.
- *All*: positions were computed using all the epochs accepted, including autonomous solutions.

Auto Topo Survey

To configure the parameters for auto-storing positions in the dynamic survey:

1. From the **Solution Type** drop-down list, select the solution type for each position computation as required. More...
2. From the *Method* drop-down list, select the method for setting the interval between the received epochs: by time, by horizontal distance, or by slope distance.
3. In the *Interval* field enter the value of this interval.

Stake Settings 1

To configure parameters that will be used during a GPS stakeout:

1. In the **Hz Dist Tolerance** edit field, enter the horizontal distance tolerance value. This is used to determine when you are close enough to the point for the bull's eye to show up.
2. From the **Reference Direction** drop-down list, select the reference direction that will be used in point stakeout. It can be North, moving direction, or the direction to the reference point (azimuth).
3. Click **Next** to configure survey parameters for staking. More...
4. Select the *Display* option from the pop-up menu that appears by clicking  in the top left corner of the dialog if required to specify an icon for the staked point. More...

Staked Point Icon

To define the icon for staked points to display on the map:

1. Select the **Use icon for staked point** box to enable settings.
2. In the **Staked Point** field:

- From the *Icon* drop-down list select the icon appearance that will be reflected it in the window.
- In the *Color* field press the ... button to set the color for the icon in the dialog. More...

Color Palette

This palette allows you to select the color for the objects to be displayed on the map.

To do this:

1. Click on the box of the desired color to highlight it.
2. Click the ok button to return to the Staked Point Icon dialog to view the colored icon.

Stake Settings 2

The *Store Staked Point As* fields allows you to set the parameters for storing staked points:

- The **Point** name can be set to the design point, next point, design point with a predefined prefix, design point with a predefined suffix. More...
- The **Note** can be set to the design point, design point with a prefix or design point with a suffix. More...

Defining names for the staked points

The Point name can be set to either design point name, next point name, design point with a pre-defined prefix (that is, stk_01, where "stk_" is the prefix), or design point with a pre-defined suffix. The choice of the prefix or suffix appears only when the corresponding item is chosen from the drop-down menu. Also, a specified numerical constant can be added to automatically generate the staked point name. For instance, if the constant specified is 1000, and the design point is 100, the staked point would be named 1100 (that is, 100+1000). If the design point is alphanumeric, the constant is appended to the name. For example, for the design point ALPHA, the corresponding staked out point is named ALPHA1000.

Setting notes for the staked points

The Note can be set to either design point name, design point with a prefix, or design point with a suffix. Also, it can be Station & Offset information. If the Station & Offset option is activated, an edit box for entering an alphanumeric prefix appears. For the United States, this prefix is "Sta", for the international markets the prefix is "Cha", and for the Korean/Japanese markets the prefix is "No". With this option activated, depending on the choice for the prefix, MAGNET Field Site automatically generates one note for each stakeout point: Sta5+5.5R5.0, Cha505.5R5.0, or No.5+5.5R5.0 respectively.

Advanced

This dialog allows you to change advanced signal processing parameters:

- Select the **Satellite System** to use from the default *GPS and GLONASS* to *GPS* only.
- From **RTK Position**, select the method of RTK corrections definition: *Extrapolation* or *Matched Epoch* (sometimes described as asynchronous or synchronous, respectively).
- By default the **Multipath Reduction** is selected to enable the use of a special signal processing technique for reduction of C/A code phase multipath.
- Select the **Canopy Environment** check box to allow the RTK engine to use less rigid thresholds when filtering out measurement outliers. This mode is recommended when working under tree canopy or in other cases of high multipath.
- Select **High-Vibration Environment (QLL)** to enable Quartz Lock Loop technology to minimize the vibration-induced impact on acquisition and tracking capabilities of the TPS receiver in high-vibration environment.
- From the **Base Station Make** drop-down list, select the manufacture of the Base receiver to designate Base Make (IGS Class) used by Rover receiver to account for GLONASS biases.
  - Select *Other* for the base receiver of other class that is included in the IGS list.
  - Select *Unknown* for the base receiver's class that is not included in the IGS list.
  - By default, it is set to *Automatic Detection* to override Base Make automatically detected by the Rover receiver if this information is transmitted by the Base.
  Note: This option requires Topcon receiver f/w at rover 3.4 or higher.
- Select the *RTK Settings* option from the pop-up menu that appears by clicking in the top left corner of the dialog if required to set more advanced parameters. More....
- Select the *Tracking Settings* option from the pop-up menu to change signal tracking parameters for the receiver used. More...

Miscellaneous (GPS)

The Miscellaneous dialog allows you to customize the user interface. The options can include:

- **Display coordinates after measurement**: when selected, computed coordinates are displayed automatically after a GPS measurement is performed and before the point coordinates are stored into the database.

- **Prompt for antenna height**: when selected, prompts for a height of the antenna before a point is stored.
- **Beep on storing points**: by default this is turned on to beep each time the point is stored.
- **Restart Epoch Counter if Solution changes**: when selected, counting epochs for Topo will stop if the specified solution is lost and will resume after it is found.
- **Auto-connect to server**: when selected (by default), an automatic connection to the server is established when you connect to the receiver in a Network survey. The Network tab of the Connections dialog will display the connection process. More...
- **Check signal quality**: select the check box to check the signal quality for the radio link in a Network survey. The Network tab of the Connections dialog will display the connection process. More...
- **Auto-disconnect from server**: when selected (by default), an automatic disconnection from the server is performed when you disconnect from the receiver in a Network survey.
- **Auto-disconnect from LongLINK**: when selected (by default), an automatic disconnection from the LongLINK modem is performed when you disconnect from the HiPer SR receiver in an RTK survey.

Network RTK

Network Real Time Kinematic is similar to RTK survey but implies that the Rover uses RTK correction data received from operating reference station networks to compute its position. Today's operating reference station networks are creating various sets of data.

To configure the Network RTK :

1. Name the configuration, and select the *Network RTK* type in the Survey dialog.
2. Select the desired **Corrections** type:
   - *MAGNET Relay* – to transmit/receive RTK corrections using TCP/IP over cellular data link.
   - *VRS* – to receive Virtual Reference Station data.
   - *MAC* –to use Master-Auxiliary Concept data.
   - *FKP* – to use network area corrections.
   - *Single Base* – to receive RTK corrections from a single base.
   - *External Config* – when the receiver uses an External program to configure RTK corrections.
3. Configure the Rover Receiver. As required, select one of the following protocols from the *Protocol* drop-down list.
   - *NTRIP* – select to receive RTK corrections from the Internet through NTRIP Caster. You have to obtain the user name and password for NTRIP server.
   - *TCP/IP* – select to receive RTK corrections from the Internet.
   - *CSD Data* – select to use the CSD form of data transmission to receive RTK corrections via a cellular phone.
4. On the Modem Connect dialog select the device the modem is connected to.
5. For Internet connection, enter the settings in the Modem Internet Info dialog. If needed, select the Use GPUID box to turn on the method of GPUID authorization.
6. For server connection, enter the user name and password in the Login Info dialog.
7. Enter dialup information in the Modem Dialup Info dialog if required.
8. For radios connected to the Receiver, configure the receiver port for connection and the Rover Radio parameters.

Modem Connect

To configure the modem connection:

1. Select **Receiver** if the modem is connected with the receiver.
2. Select **Controller** if the modem is connected with the controller.

Login Info

To configure the login information:

1. Enter your **User ID** and **Password** to login to the server with these credentials.
2. Optionally, click  and select **Lock Credentials** to hide the login information.

Enter Password

Enter and **Confirm Password** to automatically login on next connection.

MAGNET Relay

MAGNET Relay configuration implies Network RTK survey with transmitting/receiving RTK corrections using TCP/IP over cellular data link. The MAGNET Relay configuration can be used after you connect to Enterprise, and the Enterprise base receiver is started by an Enterprise user from your company.

To configure this survey type you can either select *My MAGNET Relay* configuration from the list or:

1. Name the configuration and select the *Network RTK* type in the Survey dialog.
2. Select *MAGNET Relay* from the **Corrections** drop-down list.
3. Configure the base receiver. More...
4. Configure the modem dialup information. More...
5. Configure the rover receiver. More...
6. Configure the rover modem connection. More...
7. Perform all the next steps to complete the configuration and click  to save it.

Generic NMEA

MAGNET Field Site supports Real Time DGPS survey on Topcon controllers with internal NMEA-standard GPS receivers (except GRS1). *Generic NMEA* configuration only implies reception of DGPS correction data from an SBAS system.

To configure this survey type:

1. Name the configuration, and select the *Real Time DGPS/NMEA* type in the Survey dialog.
2. Select *SBAS/Autonomous* in the **Corrections** drop-down list.
3. In Receiver Make, select *Generic NMEA* in the Manufacturer drop-down list.
4. Configure the Rover Receiver. The Receiver Model will be set to *Generic NMEA* and Antenna to *Unknown*.
5. Configure *SBAS/Autonomous* parameters.

Config: Beacon

To configure settings for a radio beacon source for differential GPS corrections:

1. Select the **Country** where the radio-beacon based differential service is located.
2. Select the **Station** that provides broadcasting differential corrections for the rover.
3. Select the **Beacon Corrections from BR-1** check box if required to use the beacon receiver BR-1 as a source of differential corrections for the rover.
4. Select the **Automatic Scan Mode** check box if you want to enable this mode in BR-1 to get the Beacon signal automatically. BR-1 will search broadcasting frequencies and output RTCM corrections from the best signal.

SBAS Setup

This dialog contains settings for the Satellite-Based Augmentation Systems (WAAS, EGNOS, or MSAS) source of differential correction data. The availability of satellite signals depends upon the receiver type and location.

To configure SBAS:

1. For GR-3 and other new receivers that support automatic tracking, you can select one of two options for use of SBAS satellites: **Best Available** or **Custom**.
   Note : If the Topcon Generic receiver was selected in the Rover Receiver dialog, custom setup is required. Not more than two SBAS satellites can be enabled for tracking in Topcon Generic receiver.
2. For custom selection, select the boxes near the PRN numbers of the satellites as required.
   Note: All satellites can be selected. The satellite most available from those selected will be used in DGPS

solution.

- PRN #, Name and Type: These columns will list all possible SBAS PRN numbers with the respective names of the satellites and types of satellite systems.
- GPS #: This PRN # applies only to the Topcon Generic receiver. One of currently unused GPS numbers should be selected in this column to be able to track this satellite in Satellite View dialog. To change, click GPS number and select the appropriate number from the pop-up menu.

3. Enable/disable use of ionospheric corrections from the SBAS satellite when computing positions. It is recommended to use ionospheric corrections.
   - *None*: ionospheric corrections are not used
   - *Apply if avail*: use ionospheric corrections if available
   - *Use sat only if avail*: use only the satellites for which ionospheric corrections are available.

Survey Configuration

Survey Configuration is a set of parameters that control a survey, define communication between devices, measure and store points. MAGNET Field contains some default configurations for different types of surveys with GNSS receivers and optical instruments. If the defaults do not suit your needs, you can create a new configuration for your survey. To do this, click ... . The Configuration dialog is displayed.

When you have created the new configuration, the name of this configuration will appear in the list.

To set a survey configuration for the job:

- From the GPS+ Configuration drop-down list of available configurations, select the required configuration.
- From the **Optical Configuration** drop-down list of available configurations, select the configuration for the Total Station or Level mode.
- Both types of configurations can be selected for one job. They will be used with the device selected upon Connections: GPS+ or Optical.
- **Hybrid Positioning** is available for selection when using any GPS+ continuous and only Robotic optical configurations for one job.
  Hybrid Positioning mode implies simultaneous connection with both GPS+ receiver and optical robot. In this mode you needn't connect/disconnect with the current device.
- Automatic Localization is available when **Hybrid Positioning mode** is activated. First **five** points will be used for calculating parameters between WGS-84 and Local coordinate systems. These parameters are saved in the job and they can be automatically updated during next measurements in the Hybrid Positioning (if residuals for these points will be less than for the previous set of points).

- After selecting the survey configurations for your job, click to store settings, which will be used each time you start a survey, and to return to the Home screen.

GPS configuration

MAGNET Field provides the set of independent wizards for creating configurations for the following GPS+ survey types:

- Network RTK
- RTK
- Network DGPS
- MAGNET Relay
- Real Time DGPS/NMEA
- PP Static
- PP Kinematic
- PP DGPS

Network RTK Survey

Network Real Time Kinematic is similar to RTK survey but it is implies that the Rover uses correction data from operating reference station networks. The rover receives the correction in the selected format and computes high accuracy coordinates on the rover side.

1. Network RTK: Configuration

To add / edit a new configuration:

1. Enter the **Name** for the configuration that will be displayed in the Configurations list.
2. In the field **Type** select the *Network RTK* configuration.
3. In the **Corrections** field select the type of correction data that will be used for survey:
    - *MAGNET Relay* – to transmit/receive RTK corrections using TCP/IP over cellular data link.
    - *VRS* – to receive Virtual Reference Station data.
    - *MAC* – to use Master-Auxiliary Concept data.
    - *FKP* – to use network area corrections.
    - *Single Base* – to receive RTK corrections from a single base.
    - *External Config* – when the receiver uses an External program to configure RTK corrections.

    If you select the *MAGNET Relay* type, you create an additional special configuration has some variation from *Network* configuration. See MAGNET Relay for the type description.

4. Optionally, you can set a value to increment/ decrement the survey point number when adding a new point.

   To do this, select the Point Properties option from the pop-up menu that displays by clicking in the top left corner.
5. Click **Next**. The wizard will open the corresponding screens to create the *Network RTK* configuration.

2. Network RTK: Receiver Make

1. If you will work with real GNSS receivers, select the vendor which developed the Rover receiver from the list in the **Rover** fields.
   If you will work without receivers, select the **Simulation Mode** check box. You can set the simulation parameters in the Simulation Setup dialog.
2. Click **Next**. The wizard will open the corresponding screen to create the *Network RTK* configuration.

3. Network RTK: Rover Receiver

To configure the rover receiver:

1. The **Ext. Receiver** box is enabled for a controller that has an internal GNSS receiver. You can select a connection with the internal GNSS receiver or any external GNSS receiver. If the controller does not have a GNSS receiver or the software is installed on the computer, the **Ext. Receiver** box is always checked. The connection for the external receivers can be *Bluetooth* and *Serial Cable*. Note: *Topcon Generic* selection will work with all receiver models which were manufactured before GR-3 receiver.
2. From the **Receiver Model** list, select the model of the rover receiver you are using and enter its **Serial Number**.
3. Set **Elevation Mask**. Usually the default 13 degrees is appropriate. Data from satellites below this elevation will not be used.
4. From **Protocol** list, select the protocol for data transmission: *TCP/IP, NTRIP, NTRIP 1.0, CSD (data call)*.
5. From the **Antenna** drop-down list, select the type of the rover receiver antenna used.
6. Enter the antenna height and set the type of height measurement (vertical or slant). To do this, click the **Antenna Height** button and select *Edit* from the drop-down menu. The Antenna Setup dialog is opened. Here you can edit the antenna type, the value of the antenna height, and the type of height.
7. If needed to transmit data from different ports of the base receiver to several rovers, you need activate the **Multiple Ports** option. To do this, click the **Peripherals** button. The Peripherals dialog is opened. Here you can select the number of ports to use for base output.
8. Optionally, you can turn off the charger mode for the receiver's internal battery if it is available. To do this, select the Receiver Settings option from the pop-up menu that displays by clicking in the top left

corner.

9. Click **Next**. The wizard will open the corresponding screen to create the *Network RTK* configuration.

4. Network RTK: Rover Modem

To configure the modem connection:

1. Select either *Receiver*, or *Controller* depending on whose modem will use for communication..
2. Click **Next**. The wizard will open the corresponding screen to create the *Network RTK* configuration.

5. Network RTK: Rover Radio

The rover radio is intended for receiving differential corrections. You can set GSM/CDMA modem (*Cellular* modem). The dialog content depends on the receiver model selected.

To configure the radio modem:

1. In the **Device Type** field select which type of Radio Modem you will use: *Internal Cellular*, or *External Cellular.*
2. In the **Device Model** field select model of the modem (for Topcon Generic, HiPer SR,HiPer II, HiPer V, GR-3,GR-5, NET G3). For HiPer SR receivers, the *Long LINK* modem does not require any additional settings.
3. For *External Cellular* specify the **Baud** rate for the **Port** to which the radio is connected and specify the **Parity**, the number of **Data** bits, the number of **Stop** bits, which are specific to the connected modem.

   Note: Sometimes during modem connection, for the "*Auto*" baud rate selection of the modem, the baud rate could not be set. The software creates an error message for this case: *" Modem baud rate could not be determined automatically. Please run TRU (see Help for details) or specify baud rate in configuration*". See here how to resolve the issue.

4. If you use an AirLink CDMA or GPRS external cellular modem, select the *Modem Register* option from the pop-up menu that displays by clicking  in the top left corner to register the radio modem. More...
5. If required, click **Defaults** to return all the communication parameters to the default settings.
6. Click **Next** to specify the Internet address as required.

6. Network RTK: Internet Address

In this dialog you can configure settings for Internet connection:

1. Enter an Internet **Address** that will be used for the connection.
2. If needed, enter a **Label** for the address which displays in the address list.

3. The desired server address you can select from the **Address list** field. To add the address to the **Address list** press the *Add New* button. To remove any address from the **Address list**, select the desired address and press the *Delete* button.
4. Select **Use GPUID**check box to send GPUID message to SAPOS Germany if required. More...
5. Click **Next** to continue customizing the Internet connection.

7. Network RTK: Login Info

In this dialog you need to enter the login information:

1. Enter a **Password** to login the selected server.
2. Optionally, you can lock / unlock the password in this dialog. To do this, select the Lock Credentials option from the pop-up menu that displays by clicking in the top left corner. If you use this option the **Password** field will display "**Locked**".
3. To unlock the password, click in the top left corner, select the Lock Credentials option and type in the password in the **Enter Password** field.
4. Click **Next** to continue customizing the internal connection.

9. Network RTK: Modem Dialup

In this dialog you can configure parameters for a dialup Internet connection:

1. Select **Provider** from the drop-down list.
2. Enter the **Dialup Number** needed to make the Internet connection.
3. Enter the **User ID** for the server.
4. Enter the **Password** to login to the server with the entered *Used ID*.
5. Enter the **PIN** number for the server
6. Enter **APN** if required.
7. To set the values to default, click the **Defaults** button.

12. Network RTK: Topo Survey

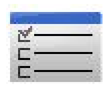
During a real-time stationary survey you can select two independent ways to save measured points to the current job: **Precise** and **Quick**. In the Topo dialog or Stake dialog, the **Precise** way will be activated by clicking , and the **Quick** way will be activated by clicking .

In the **Precise** area you can set the following parameters:

1. **Solution**: from the drop-down list, select the solution type for each position computation as required. Data will be considered only if the solution type satisfies this selection. What is displayed in the list for selection depends on the type of selected survey. More...
2. **Measure Continuously**: select the check box to log measurements continuously and stop them manually when required.
3. **Average**: the field allows you to set a specific number of measurements for logging and average them for storing the position. If needed, you can change the default number (3) of measurements.
4. **Precision**: select the check box to consider the threshold of the horizontal and vertical precisions of measurement to store the position. The default values are 0.015 m and 0.030 m, respectively.
5. **Auto Store** : select the check box to enable auto-storing positions for the average mode.

In the **Quick** area you can set the following parameters:

1. **Solution**: from the drop-down list, select the solution type for each position computation as required. Data will be considered only if the solution type satisfies this selection. What is displayed in the list for selection depends on the type of selected survey. More...
2. **Average**: the field allows you to set a specific number of measurements for logging and average them for storing the position. If required, you can change the default number (1) of measurements.
3. **Precision**: select the check box to consider the threshold of the horizontal and vertical precisions of measurement to store the position. The default values are 0.015 m and 0.030 m, respectively.

Note: Every survey parameter can be changed with the help of the  button from any in the Topo and Stake dialog in GPS+ mode.

Click **Next**. The wizard will open the corresponding screen to create the *Network RTK* configuration.

13. Network RTK: Auto Topo Survey

For for auto-storing positions in the dynamic Network RTK survey:

1. **Solution**: from the drop-down list, select the solution type for each position computation as required. More...
2. **Method** : from the drop-down list, select the method for setting the interval between the received epochs: by time, by horizontal distance, by slope distance or by elevation.
3. **Interval**: in the field, enter the value for the selected method:
   - in seconds for the *By Time* method. Default value is 1 second.
   - in meters for the *By Horiz Dist*, *By Slope Dist* and *By Elevation*. Default value is 15 meters.
4. Click **Next**. The wizard will open the corresponding screen to create the *Network RTK* configuration.

14. Network RTK: Stake Settings

To configure the parameters which will be used during a GPS stakeout:

1. In the **Horizontal Distance Tolerance** filed, enter the horizontal distance tolerance value. It is used to determine when you are close enough to the point for the bull's eye to show up.
2. In the **Screen Orientation**field you can select orientation of the screen during stakeout:
   - *North* - the top of the screen is oriented to the north during the stakeout session;
   - *Moving Direction* - the top of the screen is oriented to the direction of moving during the stakeout session;
   - *Moving Direction +North* - is similar to the *Moving Direction* option, but the screen will be oriented to the *North* direction when you are within three meters of the design point;
   - *Point/Azimuth*- if you select this orientation type, the *Orient Pt / Orient Az* field will be added to the **Stake Point** dialog. Here you can define a point or an azimuth to orient the screen top.
3. From the **Display Reference** drop-down list, select a object that will be displayed used in the *Normal View* mode.
4. Optionally, you can specify an icon for the staked point. To do this, select the Display option from the pop-up menu that displays by clicking  in the top left corner.
5. Click **Next**. The wizard will open the corresponding screen to create the *Network RTK* configuration.

15. Network RTK: Stake Survey

During a stakeout procedure you can select two independent ways to save stake out points to the current job: **Precise** and **Quick**. In the Topo dialog or Stake dialog, the **Precise** way will be activated by clicking  , and the **Quick** way will be activated by clicking .

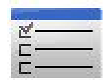
In the **Precise** area you can set the following parameters:

1. **Solution**: from the drop-down list, select the solution type for each position computation as required. Data will be considered only if the solution type satisfies this selection. What is displayed in the list for selection depends on the type of selected survey. More...
2. **Measure Continuously**: select the check box to log measurements continuously and stop them manually when required.
3. **Average**: the field allows you to set a specific number of measurements for logging and average them for storing the position. If needed, you can change the default number (3) of measurements.

4. **Precision**: select the check box to consider the threshold of the horizontal and vertical precisions of measurement to store the position. The default values are 0.015 m and 0.030 m, respectively.
5. **Auto Store** : select the check box to enable auto-storing positions for the average mode.

In the **Quick** area you can set the following parameters:

1. **Solution**: from the drop-down list, select the solution type for each position computation as required. Data will be considered only if the solution type satisfies this selection. What is displayed in the list for selection depends on the type of selected survey. More...
2. **Average**: the field allows you to set a specific number of measurements for logging and average them for storing the position. If required, you can change the default number (1) of measurements.
3. **Precision**: select the check box to consider the threshold of the horizontal and vertical precisions of measurement to store the position. The default values are 0.015 m and 0.030 m, respectively.

Note: Every survey parameter can be changed with the help of the  button from any in the Topo and Stake dialog in GPS+ mode.

Click **Next**. The wizard will open the corresponding screen to create the *Network RTK* configuration.

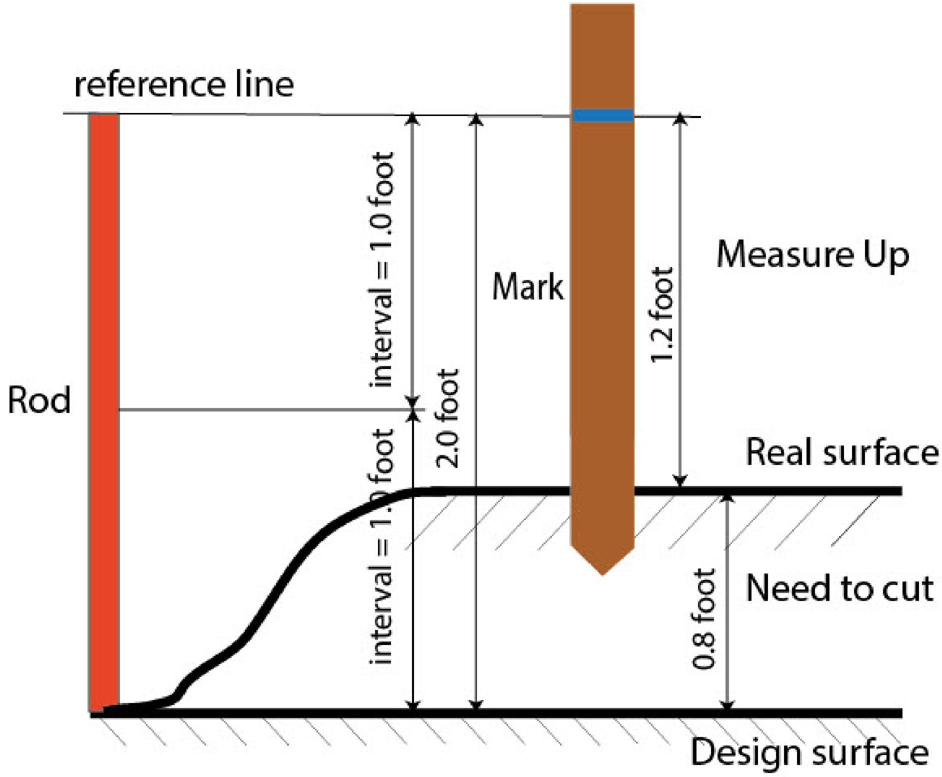
16. Network RTK: Grade Stake Marking

This dialog allows you to mark the grade stake with the skyward rounding of the cut/fill decimal values to an even number of feet (or meters).

To configure the settings for marking:

1. In the **Working Stake Length** field, enter the working length of the stake.
2. In **Top Stake Spacing** field, enter a value of top reserve part of the stake for marking.
3. In **Bottom Stake Spacing** field, enter a value of bottom reserve part of the stake for marking.
4. In **Cut/Fill interval** field, enter an even number you want the cut/fill to round to.
5. If all requirements are met, clicking **Next** opens the next **Stake Settings** dialog. More..

For instance if the cut is 0.8 and the bottom spacing is 0.5, we will round up to a cut of 2.0 (assuming 1.0 interval) and place the mark at 1.2 above ground:



17. Network RTK: Stake Settings

In the *Store Staked Point As* fields you can set the parameters for storing staked points:

1. In the **Point** field, select a method to set the name for the first staked point. The initial point name can be set to:
    - *Design point* name.
    - *Next point* name.
    - *Design point with a pre-defined prefix* (that is, stk_01, where "stk_" is the prefix).
    - *Design point with a pre-defined suffix*. The choice of the default prefix or suffix appears only when the corresponding item is chosen from the drop-down menu.
    - *Design Point plus Constant*. A specified numerical constant can be added to automatically generate the staked point name. For instance, if the constant specified is 1000, and the design point is 100, the staked point would be named 1100 (that is, 100+1000). If the design point is alphanumeric, the constant is appended to the name. For example, for the design point ALPHA, the corresponding staked out point is named ALPHA1000.
    - *Range Start*. Any start value of a range can be selected.
2. The **Note** can be set to either *Design point*name, *Design point with a prefix*, or *Design point with a suffix*. Also, it can be *Station & Offset* information. If the *Station & Offset* option is activated, an edit box for entering an alphanumeric prefix appears. For the United States, this prefix is "*Sta*", for the international markets the prefix is "*Cha*", and for the Korean/Japanese markets the prefix is "*No*". With this option activated,

depending on the choice for the prefix, MAGNET Field automatically generates one note for each stakeout point: Sta5+5.5R5.0, Cha505.5R5.0, or No.5+5.5R5.0 respectively.

18. Network RTK: Tracking

Select one of the following checkboxes to configure the satellites tracking in the receiver:

- Select **Track BDS Satellites** — to track BeiDou satellites.
- Select **Track IOV Satellites** — to track Galileo satellites.
- Select **Track SBAS Satellites** — to track SBAS satellites.
- Select **Track QZSS Satellites** — to track QZSS satellites.

Select one of the following checkboxes to configure the signal tracking for satellites::

- Select **Track L5 Signal** — to track GPS L5 and QZSS L5 signals.
- Select **Track L2C Signal** — to track GPS L2C, GLONASS C/A L2, QZSS L2C and BeiDou B2 signals.
- Select **Track GLO P Signal** — to track GLONASS P-codes on L1 and L2.

From the *Satellite System* list you can select a satellite constellation to be used for position computation for Standalone, DGPS and RTK solutions:

- *ALL* :
  - GPS satellites with L1C/A, L1P and L2P signals, plus:
    - L5 signals, if the **Track L5 Signal** check box is selected;
    - L2C signals, if the **Track L2C Signal** check box is selected.
  - GLONASS satellites with L1C/A, L1P signals, plus:
    - L2C signals, if the **Track L2C Signal** check box is selected;
    - L2P signals, if the check box **Track GLO L2P Signal** is selected.
  - If the **Track BDS Satellites** checkbox is selected — BeiDou satellites will be used. By default, the B1 signals are available. If the **Track L2C Signal** checkbox is selected, the B2 signals are available.
  - If the **Track QZSS Satellites** checkbox is selected — QZSS satellites will be used. By default, the L1C/A signals are available. If the **Track L2C Signal** checkbox is selected, the L2C signals are available.
  - If the **Track SBAS Satellites** checkbox is selected — SBAS satellites will be used. By default, the L1C/A signals are available.
- *GPS* - GPS satellites with L1C/A, L1P and L2P signals, plus
  - L5 signals, if the **Track L5 Signal** check box is selected;
  - L2C signals, if the **Track L2C Signal** check box is selected.
- *GPS + GLONASS*:
  - GPS satellites with L1C/A, L1P and L2P signals, plus
    - L5 signals, if the **Track L5 Signal**check box is selected;
    - L2C signals, if the **Track L2C Signal** check box is selected;
  - GLONASS satellites with L1C/A, L1P signals, plus

        - L2C signals, if the **Track L2C Signal** check box is selected;
        - L2P signals, if the **Track GLO L2P Signal** check box is selected.
- *GPS + BDS*:
    - GPS satellites with L1C/A, L1P and L2P signals, plus
        - L5 signals, if the **Track L5 Signal**check box is selected;
        - L2C signals, if the **Track L2C Signal** check box is selected;
    - If the **Track BDS Satellites** checkbox is selected — BeiDou satellites will be used. By default, the B1 signals are available. If the **Track L2C Signal** checkbox is selected, the B2 signals are available.
- *GPS + GLONASS + BDS*:
    - GPS satellites with L1C/A, L1P and L2P signals, plus
        - L5 signals, if the **Track L5 Signal**check box is selected;
        - L2C signals, if the **Track L2C Signal** check box is selected;
    - GLONASS satellites with L1C/A, L1P signals, plus
        - L2C signals, if the **Track L2C Signal** check box is selected;
        - L2P signals, if the **Track GLO L2P Signal** check box is selected.
    - If the **Track BDS Satellites** checkbox is selected — BeiDou satellites will be used. By default, the B1 signals are available. If the **Track L2C Signal** checkbox is selected, the B2 signals are available.

Note: The selected configuration for tracking and positioning will work properly if you're using the hardware and firmware which supports the selected signals.

19. Network RTK: Advanced

In this dialog you can select the signal processing parameters and RTK setting:

1. From **RTK Position**, select the method of RTK corrections definition:
    - *Extrapolation*(sometimes described as asynchronous) - the RTK engine will extrapolate the base station's carrier phase measurements to the current epoch (note that the truth carrier phases measured at the base cannot be transmitted and received at the rover instantly). The final positioning accuracy may be somewhat lower due to additional extrapolation errors, which may be up to a few millimeters vertical and horizontal for a one second extrapolation time.

      or

    - *Matched Epoch* (sometimes described as synchronous) - the RTK engine does not extrapolate the base station's carrier phases in position computation. Instead, the engine will either compute a delayed position or simply output the current stand-alone position (while waiting for new RTCM/CMR messages from the base station). Note that the delayed position is computed for the time (epoch) to which the last received base station's carrier phase measurements correspond. Accuracies achievable in delay mode are normally on a level with those of post-processing kinematic.

2. By default the **Multipath Reduction** is selected to enable the use of a special signal processing technique for reduction of C/A code phase multipath and C/A carrier phase multipath. This option is useful for collecting raw data near from metallic objects, or trees and high buildings.
3. Select the **Canopy Environment** check box to allow the RTK engine to use less rigid thresholds when filtering out measurement outliers. This mode is recommended when working under tree canopy or in other cases of high multipath.
4. If the GNSS receiver is collecting raw data in high-vibration environment, we recommend to select **High-Vibration Environment (QLL)** to enable Quartz Lock Loop technology to minimize the vibration-induced impact on acquisition and tracking capabilities of the TPS receiver.
5. From the **Base Station Make** drop-down list, select the manufacturer of the Base receiver to designate Base Make (IGS Class) used by the rover receiver to account for GLONASS biases.
   - By default, it is set to *Automatic Detection* to override Base Make automatically detected by the Rover receiver when this information is transmitted by the Base. The message 1033 of the RTCM format contains the information.
   - If the Base station does not transmit the manufacturer name of the Base receiver, you need to select the corresponding manufacturer name from the drop-down list.

     Note: Incorrect name of the Base receiver manufacturer can result to Float solution.
6. Optionally, you can select the settings of the RTK engine. To do this, select the RTK Settings option from the pop-up menu that displays by clicking  in the top left corner.
7. Click **Next**. The wizard will open the last screen to create the *Network RTK* configuration.

20. Network RTK: Miscellaneous

In this dialog you can you to customize the user interface:

1. **Display coordinates after measurement**: when selected, computed coordinates are displayed automatically after a GPS measurement is performed and before the point coordinates are stored into the database.
2. **Prompt for antenna height**: when selected, prompts for a height of the antenna before a point is stored.
3. **Beep on storing points**: by default this is turned on to beep each time the point is stored.
4. **Get signal quality and operator name**: when selected, the signal quality indicator and operator name will be displayed in the *Network* tab of the *Connections* dialog.
5. **Auto-disconnect from server**: when selected (by default), an automatic disconnection from the server is performed when you disconnect from the receiver.
6. **Restart Epoch Counter if Solution changes**: when selected , counting epochs for will stop if the specified solution is lost and will resume after it is found.

RTK Survey

Real-Time Kinematic survey implies usage of a pair of receivers operating simultaneously and a radio link established between the two receivers. From a functional point of view, the two receivers will differ from each other. One of the receivers (a Base) is located at a fixed point with known coordinates. The base receiver will transmit the correction data via a radio link to the rover receiver to compute high accuracy coordinates on the rover side.

1. RTK: Configuration

To add / edit a new configuration:

1. Enter the **Name** for the configuration that will be displayed in the Configurations list.
2. In the field **Type** select the *RTK* configuration.
3. Optionally, you can set a value to increment/ decrement the survey point number when adding a new point.

   To do this, select the Point Properties option from the pop-up menu that displays by clicking  in the top left corner.
4. Click **Next**. The wizard will open the corresponding screen to create the *RTK* configuration.

2. RTK: Receiver Make

1. If you will work with real GNSS receivers, select the vendor which developed the Base receiver and Rover receiver from the list in the **Base** and **Rover** fields.
   If you will work without receivers, select the **Simulation Mode** check box. You can set the simulation parameters in the Simulation Setup dialog.
2. Click **Next**. The wizard will open the corresponding screen to create the *RTK* configuration.

3. RTK: Base Receiver

To configure the base receiver:

1. The **Ext. Receiver** box is enabled for a controller that has an internal GNSS receiver. You can select a connection with the internal GNSS receiver or any external GNSS receiver. If the controller does not have a GNSS receiver or the software is installed on the computer, the **Ext. Receiver** box is always checked. The connection for the external receivers can be *Bluetooth* and *Serial Cable*.
2. From the **Receiver Model** list, select the model of the base receiver you are using and enter its **Serial Number**.
3. Set **Elevation Mask**. Usually the default 13 degrees is appropriate. Data from satellites below this elevation will not be used.
4. From **RTK Format** list, select the format for the differential corrections data, which the base radio will transmit. RTCM MSM format is supported in Topcon GNSS receivers with firmware version 4.5 and later. This format allows you to create the correction data with measurements of GPS, GLONASS and BeiDou satellite systems.

5. From the **Antenna** drop-down list, select the type of the base receiver antenna used.
6. Enter the antenna height and set the type of height measurement (vertical or slant). To do this, click the **Antenna Height** button and select *Edit* from the drop-down menu. The Antenna Setup dialog is opened. Here you can edit the antenna type, the value of the antenna height, and the type of height.
7. If needed to transmit data from different ports of the base receiver to several rovers, you need activate the **Multiple Ports** option. To do this, click the **Peripherals** button. The Peripherals dialog is opened. Here you can select the number of ports to use for base output.
8. Optionally, you can turn off the charger mode for the receiver's internal battery if it is available. To do this, select the Receiver Settings option from the pop-up menu that displays by clicking  in the top left corner.
9. Click **Next**. The wizard will open the corresponding screen to create the *RTK* configuration.

5. RTK: Base Radio

The base radio is intended for transmitting differential corrections. You can set GSM/CDMA modem (*Cellular* modem) or UHF modem (*Radio*) or *LongLINK* for HiPer SR receiver. The dialog content depends on the receiver model selected.

To configure the radio modem:

1. In the **Device Type** field select which type of Radio Modem you will use: *Internal Radio*, or *Internal Cellular*, or *External Radio*, or *External Cellular* (for Topcon Generic, HiPer II, HiPer V, GR-3,GR-5, NET G3).
2. In the **Device Model** field select model of the modem (for Topcon Generic, HiPer SR,HiPer II, HiPer V, GR-3,GR-5, NET G3). For HiPer SR receivers, the *Long LINK* modem does not require any additional settings.
3. For *Internal Radio*, *External Radio*, *External Cellular* specify the **Baud** rate for the receiver **Port** that the radio is connected to.

   Note: Sometimes during modem connection, for the "*Auto*" baud rate selection of the modem, the baud rate could not be set. The software creates an error message for this case: *" Modem baud rate could not be determined automatically. Please run TRU (see Help for details) or specify baud rate in configuration*". See here how to resolve the issue.

4. For *External Radio*, *External Cellular* specify communication parameters for the receiver **Port** that the radio is connected to: **Parity**, the number of **Data** bits, the number of **Stop** bits, which are specific to the connected modem.
5. If you use an AirLink CDMA or GPRS external cellular modem, select the *Modem Register* option from the pop-up menu that displays by clicking  in the top left corner to register the radio modem. More...

6. If required, click **Defaults** to return all the communication parameters to the default settings.
7. Click **Next** to specify the base and rover modem radio parameters as required.

RTK: Base and Rover Radio Parameters

The type of the dialog is dependent upon the selected modem type for the base or radio receiver. Click the desired modem type to open an instruction how to configure modem parameters:

1. Internal Radio:
   - Digital UHF / Digital UHFII
   - FH 915 Plus
   - Satel
2. Internal Cellular
   - Auto, Digital UHF I/II GSM,FH915 + GSM, General Internal GSM, Satel GSM,
   - Digital UHF CDMA
   - TCP/IP
3. External Radio
   - RE-S1
   - Satel, SRL-35
   - TR-35
4. External Cellular
   - AirLink CDMA (MUDP) for base
   - AirLink GPRS for rover
   - AirLink CDMA for rover
   - Generic CDMA for rover
   - Generic GSM, MultiTech GSM, Siemens TC35
   - TCP/IP for base

You can set GSM/CDMA modem (*Cellular* modem) or UHF modem (*Radio*). The dialog content depends on the receiver model selected.

6. RTK: Rover Receiver

To configure the rover receiver:

1. The **Ext. Receiver** box is enabled for a controller that has an internal GNSS receiver. You can select a connection with the internal GNSS receiver or any external GNSS receiver. If the controller does not have a GNSS receiver or the software is installed on the computer, the **Ext. Receiver** box is always checked. The connection for the external receivers can be *Bluetooth* and *Serial Cable*. Note: *Topcon Generic* selection will work with all receiver models which were manufactured before GR-3 receiver.
2. From the **Receiver Model** list, select the model of the rover receiver you are using and enter its **Serial Number**.

3. Set **Elevation Mask**. Usually the default 13 degrees is appropriate. Data from satellites below this elevation will not be used.
4. From **RTK Format** list, select the format for the differential corrections data, which the rover radio modem will receive.
5. From the **Antenna** drop-down list, select the type of the rover receiver antenna used.
6. Enter the antenna height and set the type of height measurement (vertical or slant). To do this, click the **Antenna Height** button and select *Edit* from the drop-down menu. The Antenna Setup dialog is opened. Here you can edit the antenna type, the value of the antenna height, and the type of height.
7. If needed to transmit data from different ports of the base receiver to several rovers, you need activate the **Multiple Ports** option. To do this, click the **Peripherals** button. The Peripherals dialog is opened. Here you can select the number of ports to use for base output.
8. Optionally, you can turn off the charger mode for the receiver's internal battery if it is available. To do this, select the Receiver Settings option from the pop-up menu that displays by clicking  in the top left corner.
9. Click **Next**. The wizard will open the corresponding screen to create the *RTK* configuration.

7. RTK: Rover Modem

To configure the modem connection:

1. Select either *Receiver*, or *Controller* depending on whose modem will use for communication.
2. Click **Next**. The wizard will open the corresponding screen to create the *RTK* configuration.

8. RTK: Rover Radio

The rover radio is intended for receiving differential corrections. You can set GSM/CDMA modem (*Cellular* modem) or UHF modem (*Radio*). The dialog content depends on the receiver model selected.

To configure the radio modem:

1. In the **Device Type** field select which type of Radio Modem you will use: *Internal Radio*, or *Internal Cellular*, or *External Radio*, or *External Cellular* (for Topcon Generic, HiPer II, HiPer V, GR-3,GR-5, NET G3).
2. In the **Device Model** field select model of the modem (for Topcon Generic, HiPer SR,HiPer II, HiPer V, GR-3,GR-5, NET G3). For HiPer SR receivers, the *Long LINK* modem does not require any additional settings.
3. For *Internal Radio*, *External Radio*, *External Cellular* specify the **Baud** rate for the **Port** to which the radio is connected.

   Note: Sometimes during modem connection, for the "*Auto*" baud rate selection of the modem, the baud rate could not be set. The software creates an error message for this case: *” Modem baud rate could not be* *determined automatically. Please run TRU (see Help for details) or specify baud rate in configuration*". See here how to resolve the issue.

4. For *External Radio*, *External Cellular* specify communication parameters for the receiver **Port** that the radio is connected to: **Parity**, the number of **Data** bits, the number of **Stop** bits, which are specific to the connected modem.
5. If you use an AirLink CDMA or GPRS external cellular modem, select the *Modem Register* option from the pop-up menu that displays by clicking  in the top left corner to register the radio modem. More...
6. If required, click **Defaults** to return all the communication parameters to the default settings.
7. Click **Next** to specify the base and rover modem radio parameters as required.

11. RTK: Topo Survey

During a real-time stationary survey you can select two independent ways to save measured points to the current job: **Precise** and **Quick**. In the Topo dialog or Stake dialog, the **Precise** way will be activated by clicking , and the **Quick** way will be activated by clicking .

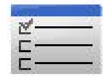
In the **Precise** area you can set the following parameters:

1. **Solution**: from the drop-down list, select the solution type for each position computation as required. Data will be considered only if the solution type satisfies this selection. What is displayed in the list for selection depends on the type of selected survey. More...
2. **Measure Continuously**: select the check box to log measurements continuously and stop them manually when required.
3. **Average**: the field allows you to set a specific number of measurements for logging and average them for storing the position. If needed, you can change the default number (3) of measurements.
4. **Precision**: select the check box to consider the threshold of the horizontal and vertical precisions of measurement to store the position. The default values are 0.015 m and 0.030 m, respectively.
5. **Auto Store** : select the check box to enable auto-storing positions for the average mode.

In the **Quick** area you can set the following parameters:

1. **Solution**: from the drop-down list, select the solution type for each position computation as required. Data will be considered only if the solution type satisfies this selection. What is displayed in the list for selection depends on the type of selected survey. More...

2. **Average**: the field allows you to set a specific number of measurements for logging and average them for storing the position. If required, you can change the default number (1) of measurements.
3. **Precision**: select the check box to consider the threshold of the horizontal and vertical precisions of measurement to store the position. The default values are 0.015 m and 0.030 m, respectively.

Note: Every survey parameter can be changed with the help of the  button from any in the Topo and Stake dialog in GPS+ mode.

Click **Next**. The wizard will open the corresponding screen to create the *RTK* configuration.

12. RTK: Auto Topo Survey

For for auto-storing positions in the dynamic RTK survey:

1. **Solution**: from the drop-down list, select the solution type for each position computation as required. More...
2. **Method** : from the drop-down list, select the method for setting the interval between the received epochs: by time, by horizontal distance, by slope distance or by elevation.
3. **Interval**: in the field, enter the value for the selected method:
   - in seconds for the *By Time* method. Default value is 1 second.
   - in meters for the *By Horiz Dist*, *By Slope Dist* and *By Elevation*. Default value is 1 meter.
4. Click **Next**. The wizard will open the corresponding screen to create the *RTK* configuration.

13. RTK: Stake Settings

To configure the parameters which will be used during a GPS stakeout:

1. In the **Horizontal Distance Tolerance** filed, enter the horizontal distance tolerance value. It is used to determine when you are close enough to the point for the bull's eye to show up.
2. In the **Screen Orientation**field you can select orientation of the screen during stakeout:
   - *North* - the top of the screen is oriented to the north during the stakeout session;
   - *Moving Direction* - the top of the screen is oriented to the direction of moving during the stakeout session;
   - *Moving Direction +North* - is similar to the *Moving Direction* option, but the screen will be oriented to the *North* direction when you are within three meters of the design point;
   - *Point/Azimuth*- if you select this orientation type, the *Orient Pt / Orient Az* field will be added to the **Stake Point** dialog. Here you can define a point or an azimuth to orient the screen top.
3. From the **Display Reference** drop-down list, select a object that will be displayed used in the *Normal View* mode.
4. Optionally, you can specify an icon for the staked point. To do this, select the Display option from the pop-up

menu that displays by clicking in the top left corner.

5. Click **Next**. The wizard will open the corresponding screen to create the *RTK* configuration.

14. RTK: Stake Survey

During a stakeout procedure you can select two independent ways to save stake out points to the current job: **Precise** and **Quick**. In the Topo dialog or Stake dialog, the **Precise** way will be activated by clicking , and the **Quick** way will be activated by clicking .

In the **Precise** area you can set the following parameters:

1. **Solution**: from the drop-down list, select the solution type for each position computation as required. Data will be considered only if the solution type satisfies this selection. What is displayed in the list for selection depends on the type of selected survey. More...
2. **Measure Continuously**: select the check box to log measurements continuously and stop them manually when required.
3. **Average**: the field allows you to set a specific number of measurements for logging and average them for storing the position. If needed, you can change the default number (3) of measurements.
4. **Precision**: select the check box to consider the threshold of the horizontal and vertical precisions of measurement to store the position. The default values are 0.015 m and 0.030 m, respectively.
5. **Auto Store** : select the check box to enable auto-storing positions for the average mode.

In the **Quick** area you can set the following parameters:

1. **Solution**: from the drop-down list, select the solution type for each position computation as required. Data will be considered only if the solution type satisfies this selection. What is displayed in the list for selection depends on the type of selected survey. More...
2. **Average**: the field allows you to set a specific number of measurements for logging and average them for storing the position. If required, you can change the default number (1) of measurements.
3. **Precision**: select the check box to consider the threshold of the horizontal and vertical precisions of measurement to store the position. The default values are 0.015 m and 0.030 m, respectively.

Note: Every survey parameter can be changed with the help of the button from any in the Topo and Stake dialog in GPS+ mode.

Click **Next**. The wizard will open the corresponding screen to create the *RTK* configuration.

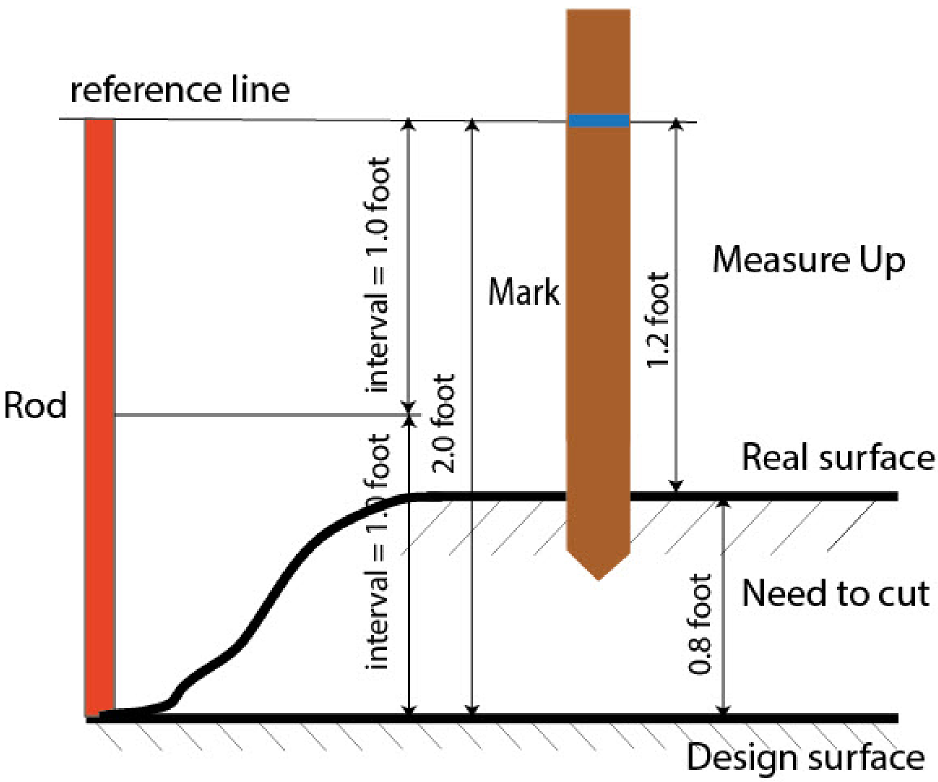
15. RTK: Grade Stake Marking

This dialog allows you to mark the grade stake with the skyward rounding of the cut/fill decimal values to an even number of feet (or meters).

To configure the settings for marking:

1. In the **Working Stake Length** field, enter the working length of the stake.
2. In **Top Stake Spacing** field, enter a value of top reserve part of the stake for marking.
3. In **Bottom Stake Spacing** field, enter a value of bottom reserve part of the stake for marking.
4. In **Cut/Fill interval** field, enter an even number you want the cut/fill to round to.
5. If all requirements are met, clicking **Next** opens the next **Stake Settings** dialog. More..

For instance if the cut is 0.8 and the bottom spacing is 0.5, we will round up to a cut of 2.0 (assuming 1.0 interval) and place the mark at 1.2 above ground:



16. RTK: Stake Settings

In the *Store Staked Point As* fields you can set the parameters for storing staked points:

1. In the **Point** field, select a method to set the name for the first staked point. The initial point name can be set to:

- *Design point* name.
- *Next point* name.
- *Design point with a pre-defined prefix* (that is, stk_01, where “stk_” is the prefix).
- *Design point with a pre-defined suffix*. The choice of the default prefix or suffix appears only when the corresponding item is chosen from the drop-down menu.
- *Design Point plus Constant*. A specified numerical constant can be added to automatically generate the staked point name. For instance, if the constant specified is 1000, and the design point is 100, the staked point would be named 1100 (that is, 100+1000). If the design point is alphanumeric, the constant is appended to the name. For example, for the design point ALPHA, the corresponding staked out point is named ALPHA1000.
- *Range Start*. Any start value of a range can be selected.

2. The **Note** can be set to either *Design point*name, *Design point with a prefix*, or *Design point with a suffix*. Also, it can be *Station & Offset* information. If the *Station & Offset* option is activated, an edit box for entering an alphanumeric prefix appears. For the United States, this prefix is “*Sta*”, for the international markets the prefix is “*Cha*”, and for the Korean/Japanese markets the prefix is “*No*”. With this option activated, depending on the choice for the prefix, MAGNET Field automatically generates one note for each stakeout point: Sta5+5.5R5.0, Cha505.5R5.0, or No.5+5.5R5.0 respectively.

17. RTK: Tracking

Select one of the following checkboxes to configure the satellites tracking in the receiver:

- Select **Track BDS Satellites** — to track BeiDou satellites.
- Select **Track IOV Satellites** — to track Galileo satellites.
- Select **Track SBAS Satellites** — to track SBAS satellites.
- Select **Track QZSS Satellites** — to track QZSS satellites.

Select one of the following checkboxes to configure the signal tracking for satellites::

- Select **Track L5 Signal** — to track GPS L5 and QZSS L5 signals.
- Select **Track L2C Signal** — to track GPS L2C, GLONASS C/A L2, QZSS L2C and BeiDou B2 signals.
- Select **Track GLO P Signal** — to track GLONASS P-codes on L1 and L2.

From the *Satellite System* list you can select a satellite constellation to be used for position computation for Standalone, DGPS and RTK solutions:

- *ALL* :
  - GPS satellites with L1C/A, L1P and L2P signals, plus:
    - L5 signals, if the **Track L5 Signal** check box is selected;
    - L2C signals, if the **Track L2C Signal** check box is selected.
  - GLONASS satellites with L1C/A, L1P signals, plus:

        - L2C signals, if the **Track L2C Signal** check box is selected;
        - L2P signals, if the check box **Track GLO L2P Signal** is selected.
    - If the **Track BDS Satellites** checkbox is selected — BeiDou satellites will be used. By default, the B1 signals are available. If the **Track L2C Signal** checkbox is selected, the B2 signals are available.
    - If the **Track QZSS Satellites** checkbox is selected — QZSS satellites will be used. By default, the L1C/A signals are available. If the **Track L2C Signal** checkbox is selected, the L2C signals are available.
    - If the **Track SBAS Satellites** checkbox is selected — SBAS satellites will be used. By default, the L1C/A signals are available.
- *GPS* - GPS satellites with L1C/A, L1P and L2P signals, plus
    - L5 signals, if the **Track L5 Signal** check box is selected;
    - L2C signals, if the **Track L2C Signal** check box is selected.
- *GPS + GLONASS*:
    - GPS satellites with L1C/A, L1P and L2P signals, plus
        - L5 signals, if the **Track L5 Signal**check box is selected;
        - L2C signals, if the **Track L2C Signal** check box is selected;
    - GLONASS satellites with L1C/A, L1P signals, plus
        - L2C signals, if the **Track L2C Signal** check box is selected;
        - L2P signals, if the **Track GLO L2P Signal** check box is selected.
- *GPS + BDS*:
    - GPS satellites with L1C/A, L1P and L2P signals, plus
        - L5 signals, if the **Track L5 Signal**check box is selected;
        - L2C signals, if the **Track L2C Signal** check box is selected;
    - If the **Track BDS Satellites** checkbox is selected — BeiDou satellites will be used. By default, the B1 signals are available. If the **Track L2C Signal** checkbox is selected, the B2 signals are available.
- *GPS + GLONASS + BDS*:
    - GPS satellites with L1C/A, L1P and L2P signals, plus
        - L5 signals, if the **Track L5 Signal**check box is selected;
        - L2C signals, if the **Track L2C Signal** check box is selected;
    - GLONASS satellites with L1C/A, L1P signals, plus
        - L2C signals, if the **Track L2C Signal** check box is selected;
        - L2P signals, if the **Track GLO L2P Signal** check box is selected.
    - If the **Track BDS Satellites** checkbox is selected — BeiDou satellites will be used. By default, the B1 signals are available. If the **Track L2C Signal** checkbox is selected, the B2 signals are available.

Note: The selected configuration for tracking and positioning will work properly if you're using the hardware and firmware which supports the selected signals.

18. RTK: Advanced

In this dialog you can select the signal processing parameters and RTK setting:

1. From **RTK Position**, select the method of RTK corrections definition:
   - *Extrapolation*(sometimes described as asynchronous) - the RTK engine will extrapolate the base station's carrier phase measurements to the current epoch (note that the truth carrier phases measured at the base cannot be transmitted and received at the rover instantly). The final positioning accuracy may be somewhat lower due to additional extrapolation errors, which may be up to a few millimeters vertical and horizontal for a one second extrapolation time.

     or

   - *Matched Epoch* (sometimes described as synchronous) - the RTK engine does not extrapolate the base station's carrier phases in position computation. Instead, the engine will either compute a delayed position or simply output the current stand-alone position (while waiting for new RTCM/CMR messages from the base station). Note that the delayed position is computed for the time (epoch) to which the last received base station's carrier phase measurements correspond. Accuracies achievable in delay mode are normally on a level with those of post-processing kinematic.
2. By default the **Multipath Reduction** is selected to enable the use of a special signal processing technique for reduction of C/A code phase multipath and C/A carrier phase multipath. This option is useful for collecting raw data near from metallic objects, or trees and high buildings.
3. Select the **Canopy Environment** check box to allow the RTK engine to use less rigid thresholds when filtering out measurement outliers. This mode is recommended when working under tree canopy or in other cases of high multipath.
4. If the GNSS receiver is collecting raw data in high-vibration environment, we recommend to select **High-Vibration Environment (QLL)** to enable Quartz Lock Loop technology to minimize the vibration-induced impact on acquisition and tracking capabilities of the TPS receiver.
5. From the **Base Station Make** drop-down list, select the manufacturer of the Base receiver to designate Base Make (IGS Class) used by the rover receiver to account for GLONASS biases.
   - By default, it is set to *Automatic Detection* to override Base Make automatically detected by the Rover receiver when this information is transmitted by the Base. The message 1033 of the RTCM format contains the information.
   - If the Base station does not transmit the manufacturer name of the Base receiver, you need to select the corresponding manufacturer name from the drop-down list.

     Note: Incorrect name of the Base receiver manufacturer can result to Float solution.
6. Optionally, you can select the settings of the RTK engine. To do this, select the RTK Settings option from the pop-up menu that displays by clicking  in the top left corner.
7. Click **Next**. The wizard will open the last screen to create the *RTK* configuration.

19. RTK: Miscellaneous

In this dialog you can you to customize the user interface:

1. **Display coordinates after measurement**: when selected, computed coordinates are displayed automatically after a GPS measurement is performed and before the point coordinates are stored into the database.
2. **Prompt for antenna height**: when selected, prompts for a height of the antenna before a point is stored.
3. **Beep on storing points**: by default this is turned on to beep each time the point is stored.
4. **Restart Epoch Counter if Solution changes**: when selected , counting epochs for will stop if the specified solution is lost and will resume after it is found.
5. **Auto-disconnect from LongLINK**: when selected (by default), an automatic disconnection from the LongLINK modem is performed when you disconnect from the HiPer SR receiver in an RTK survey.

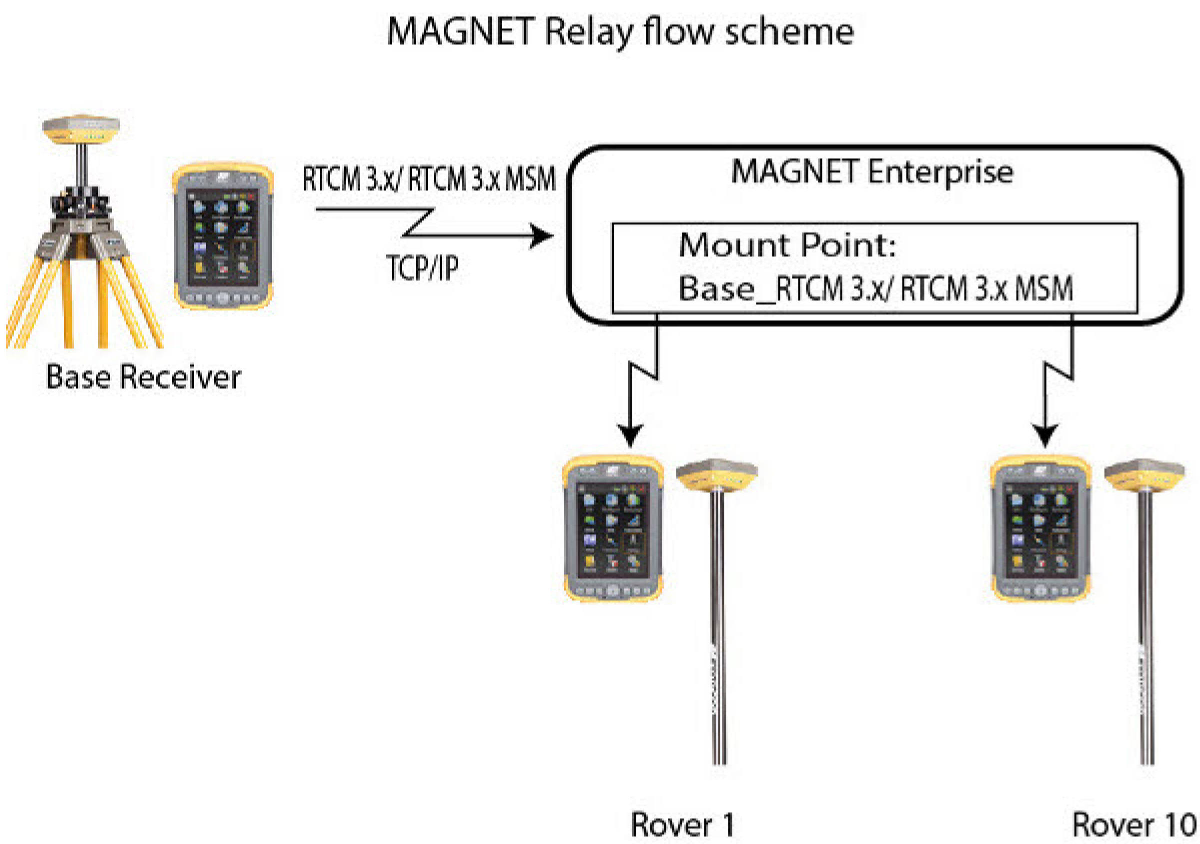
MAGNET Relay

MAGNET Relay configuration allows you to use **any** Topcon GNSS receiver as the base receiver for up to ten rover receivers. There's no need to use a SIM card with fixed IP for the base receiver in such configuration. Moreover, the base and rover receivers do not need the internal cellular modems. The communication will be provided by internal modems of the controller with MAGNET Field.

The MAGNET Relay configuration can be used after you connect to MAGNET Enterprise. To apply the configuration, you need:

- the login and password to Enterprise server
- the subscription to MAGNET Relay.

The base receiver transmits the correction data to the MAGNET Enterprise; rover receivers will be able to select active mount point and receive the corrections from the given base after connected to the Enterprise.



Before running the Base Receiver, you need to connect the controller to MAGNET Enterprise server using the internal modem of the controller. If the receiver has an internal/external cell modem and you selected the receiver modem in the configuration, MAGNET Field will automatically establish the connection of the receiver modem with MAGNET Enterprise server during **Start Base** procedure. After that you can disconnect the controller from MAGNET Enterprise server; the base receiver will continue to transmit correction data to MAGNET Enterprise server using its own modem. If the receiver does not have an internal/external cell modem, select the *Controller* modem in the configuration and the controller will transmit the correction data to MAGNET Enterprise server during whole session.

Before running the Rover Receiver, you need to connect the controller to MAGNET Enterprise server using the internal modem of the controller. We recommend to select in configuration the *Controller* modem. In this case the controller will receive the correction data from MAGNET Enterprise server during whole session.

1. MAGNET Relay: Configuration

To add / edit a new configuration:

1. Enter the **Name** for the configuration that will be displayed in the Configurations list.
2. In the field **Type** select the *Network RTK* configuration.
3. In the **Corrections** field select *MAGNET Relay*
4. Optionally, you can set a value to increment/ decrement the survey point number when adding a new point.

   To do this, select the Point Properties option from the pop-up menu that displays by clicking  in the top left corner.
5. Click **Next**. The wizard will open the corresponding screens to create the *MAGNET Relay* configuration.

2. MAGNET Relay: Receiver Make

1. If you will work with real GNSS receivers, select the vendor which developed the Base receiver and Rover receiver from the list in the **Base** and **Rover** fields.
   If you will work without receivers, select the **Simulation Mode** check box. You can set the simulation parameters in the Simulation Setup dialog.
2. Click **Next**. The wizard will open the corresponding screen to create the *MAGNET Relay* configuration.

3. MAGNET Relay: Base Receiver

To configure the base receiver:

1. The **Ext. Receiver** box is enabled for a controller that has an internal GNSS receiver. You can select a connection with the internal GNSS receiver or any external GNSS receiver. If the controller does not have a GNSS receiver or the software is installed on the computer, the **Ext. Receiver** box is always checked. The connection for the external receivers can be *Bluetooth* and *Serial Cable*.
2. From the **Receiver Model** list, select the model of the base receiver you are using and enter its **Serial Number**.
3. Set **Elevation Mask**. Usually the default 13 degrees is appropriate. Data from satellites below this elevation will not be used.
4. From **RTK Format** list, select the format for the differential corrections data, which the base radio will transmit. RTCM MSM format is supported in Topcon GNSS receivers with firmware version 4.5 and later. This format allows you to create the correction data with measurements of GPS, GLONASS and BeiDou satellite systems.

5. The **Relay Name** field will show the user's MAGNET Field license serial number as the default relay name on the given controller. This name is used when the Enterprise server generates a mount point name and the Start Base dialog. If there is no serial number yet, the *Relay* name is offered.
6. From the **Antenna** drop-down list, select the type of the base receiver antenna used.
7. Enter the antenna height and set the type of height measurement (vertical or slant). To do this, click the **Antenna Height** button and select *Edit* from the drop-down menu. The Antenna Setup dialog is opened. Here you can edit the antenna type, the value of the antenna height, and the type of height.
8. If needed to transmit data from different ports of the base receiver to several rovers, you need activate the **Multiple Ports** option. To do this, click the **Peripherals** button. The Peripherals dialog is opened. Here you can select the number of ports to use for base output.
9. Optionally, you can turn off the charger mode for the receiver's internal battery if it is available. To do this, select the Receiver Settings option from the pop-up menu that displays by clicking  in the top left corner.
10. Click **Next**. The wizard will open the corresponding screen to create the *MAGNET Relay* configuration.

4. MAGNET Relay: Base Modem

To configure the modem connection:

1. Select either *Receiver*, or *Controller* depending on whose modem will use for communication. When you select the receiver modem, MAGNET Field automatically establishes the connection of the receiver modem with MAGNET Enterprise server during **Start Base** procedure. After that you can disconnect the controller from MAGNET Enterprise server; the base receiver will continue to transmit correction data to MAGNET Enterprise server using own modem.
2. Click **Next**. The wizard will open the corresponding screen to create the *MAGNET Relay* configuration.

6. MAGNET Relay: Base Radio

The base radio is intended for transmitting corrections data to the MAGNET Enterprise server. The dialog content depends on your modem selection in the Config: Base Modem dialog:

- If you selected the receiver modem, in the **Device Type** field, select either *Internal Cellular* or *External Cellular*.
- If you selected the controller modem, the internal cellular controller modem will always be used for connection with existing network.

Click **Next**. The wizard will open the corresponding screen to create the *MAGNET Relay* configuration.

7. MAGNET Relay: Modem Dialup

In this dialog you can configure parameters for a dialup Internet connection:

1. Select **Provider** from the drop-down list.
2. Enter the **Dialup Number** needed to make the Internet connection.
3. Enter the **User ID** for the server.
4. Enter the **Password** to login to the server with the entered *Used ID*.
5. Enter the **PIN** number for the server
6. Enter **APN** if required.
7. To set the values to default, click the **Defaults** button.

8. MAGNET Relay: Rover Receiver

To configure the rover receiver:

1. The **Ext. Receiver** box is enabled for a controller that has an internal GNSS receiver. You can select a connection with the internal GNSS receiver or any external GNSS receiver. If the controller does not have a GNSS receiver or the software is installed on the computer, the **Ext. Receiver** box is always checked. The connection for the external receivers can be *Bluetooth* and *Serial Cable*. Note: *Topcon Generic* selection will work with all receiver models which were manufactured before GR-3 receiver.
2. From the **Receiver Model** list, select the model of the rover receiver you are using and enter its **Serial Number**.
3. Set **Elevation Mask**. Usually the default 13 degrees is appropriate. Data from satellites below this elevation will not be used.
4. From **Protocol** list, the *MAGNET Relay* protocol is always selected.
5. From the **Antenna** drop-down list, select the type of the rover receiver antenna used.
6. Enter the antenna height and set the type of height measurement (vertical or slant). To do this, click the **Antenna Height** button and select *Edit* from the drop-down menu. The Antenna Setup dialog is opened. Here you can edit the antenna type, the value of the antenna height, and the type of height.
7. If needed to transmit data from different ports of the base receiver to several rovers, you need activate the **Multiple Ports** option. To do this, click the **Peripherals** button. The Peripherals dialog is opened. Here you can select the number of ports to use for base output.
8. Optionally, you can turn off the charger mode for the receiver's internal battery if it is available. To do this, select the Receiver Settings option from the pop-up menu that displays by clicking in the top left corner.
9. Click **Next**. The wizard will open the corresponding screen to create the *MAGNET Relay* configuration.

9. MAGNET Relay: Rover Modem

We recommend to use the modem controller for the rover receiver. In this case the controller will receive the correction data from MAGNET Enterprise server during whole session.

Click **Next**. The wizard will open the corresponding screen to create the *MAGNET Relay* configuration.

10. MAGNET Relay: Rover Radio

The rover radio is intended for receiving differential corrections from the MAGNET Enterprise server. If you selected the controller modem in the Config: Rover Modem dialog, the internal cellular controller modem will always be used for connection with existing network.

Click **Next**. The wizard will open the corresponding screen to create the *MAGNET Relay* configuration.

.

13. MAGNET Relay: Topo Survey

During a real-time stationary survey you can select two independent ways to save measured points to the current job: **Precise** and **Quick**. In the Topo dialog or Stake dialog, the **Precise** way will be activated by clicking , and the **Quick** way will be activated by clicking .

In the **Precise** area you can set the following parameters:

1. **Solution**: from the drop-down list, select the solution type for each position computation as required. Data will be considered only if the solution type satisfies this selection. What is displayed in the list for selection depends on the type of selected survey. More...
2. **Measure Continuously**: select the check box to log measurements continuously and stop them manually when required.
3. **Average**: the field allows you to set a specific number of measurements for logging and average them for storing the position. If needed, you can change the default number (3) of measurements.
4. **Precision**: select the check box to consider the threshold of the horizontal and vertical precisions of measurement to store the position. The default values are 0.015 m and 0.030 m, respectively.
5. **Auto Store** : select the check box to enable auto-storing positions for the average mode.

In the **Quick** area you can set the following parameters:

1. **Solution**: from the drop-down list, select the solution type for each position computation as required. Data will be considered only if the solution type satisfies this selection. What is displayed in the list for selection depends on the type of selected survey. More...
2. **Average**: the field allows you to set a specific number of measurements for logging and average them for storing the position. If required, you can change the default number (1) of measurements.
3. **Precision**: select the check box to consider the threshold of the horizontal and vertical precisions of measurement to store the position. The default values are 0.015 m and 0.030 m, respectively.

Note: Every survey parameter can be changed with the help of the  button from any in the Topo and Stake dialog in GPS+ mode.

Click **Next**. The wizard will open the corresponding screen to create the *MAGNET Relay* configuration.

14. MAGNET Relay: Auto Topo Survey

For for auto-storing positions in the dynamic MAGNET Relay survey:

1. **Solution**: from the drop-down list, select the solution type for each position computation as required. More...
2. **Method** : from the drop-down list, select the method for setting the interval between the received epochs: by time, by horizontal distance, by slope distance or by elevation.
3. **Interval**: in the field, enter the value for the selected method:
   - in seconds for the *By Time* method. Default value is 1 second.
   - in meters for the *By Horiz Dist*, *By Slope Dist* and *By Elevation*. Default value is 15 meters.
4. Click **Next**. The wizard will open the corresponding screen to create the *MAGNET Relay* configuration.

15. MAGNET Relay: Stake Settings

To configure the parameters which will be used during a GPS stakeout:

1. In the **Horizontal Distance Tolerance** filed, enter the horizontal distance tolerance value. It is used to determine when you are close enough to the point for the bull's eye to show up.
2. In the **Screen Orientation**field you can select orientation of the screen during stakeout:
   - *North* - the top of the screen is oriented to the north during the stakeout session;
   - *Moving Direction* - the top of the screen is oriented to the direction of moving during the stakeout session;
   - *Moving Direction +North* - is similar to the *Moving Direction* option, but the screen will be oriented to the *North* direction when you are within three meters of the design point;
   - *Point/Azimuth*- if you select this orientation type, the *Orient Pt / Orient Az* field will be added to the **Stake Point** dialog. Here you can define a point or an azimuth to orient the screen top.
3. From the **Display Reference** drop-down list, select a object that will be displayed used in the *Normal View* mode.
4. Optionally, you can specify an icon for the staked point. To do this, select the Display option from the pop-up menu that displays by clicking  in the top left corner.
5. Click **Next**. The wizard will open the corresponding screen to create the *MAGNET Relay* configuration.

16. MAGNET Relay: Stake Survey

During a stakeout procedure you can select two independent ways to save stake out points to the current job: **Precise** and **Quick**. In the Topo dialog or Stake dialog, the **Precise** way will be activated by clicking

, and the **Quick** way will be activated by clicking .

In the **Precise** area you can set the following parameters:

1. **Solution**: from the drop-down list, select the solution type for each position computation as required. Data will be considered only if the solution type satisfies this selection. What is displayed in the list for selection depends on the type of selected survey. More...
2. **Measure Continuously**: select the check box to log measurements continuously and stop them manually when required.
3. **Average**: the field allows you to set a specific number of measurements for logging and average them for storing the position. If needed, you can change the default number (3) of measurements.
4. **Precision**: select the check box to consider the threshold of the horizontal and vertical precisions of measurement to store the position. The default values are 0.015 m and 0.030 m, respectively.
5. **Auto Store** : select the check box to enable auto-storing positions for the average mode.

In the **Quick** area you can set the following parameters:

1. **Solution**: from the drop-down list, select the solution type for each position computation as required. Data will be considered only if the solution type satisfies this selection. What is displayed in the list for selection depends on the type of selected survey. More...
2. **Average**: the field allows you to set a specific number of measurements for logging and average them for storing the position. If required, you can change the default number (1) of measurements.
3. **Precision**: select the check box to consider the threshold of the horizontal and vertical precisions of measurement to store the position. The default values are 0.015 m and 0.030 m, respectively.

Note: Every survey parameter can be changed with the help of the button from any in the Topo and Stake dialog in GPS+ mode.

Click **Next**. The wizard will open the corresponding screen to create the *MAGNET Relay* configuration.

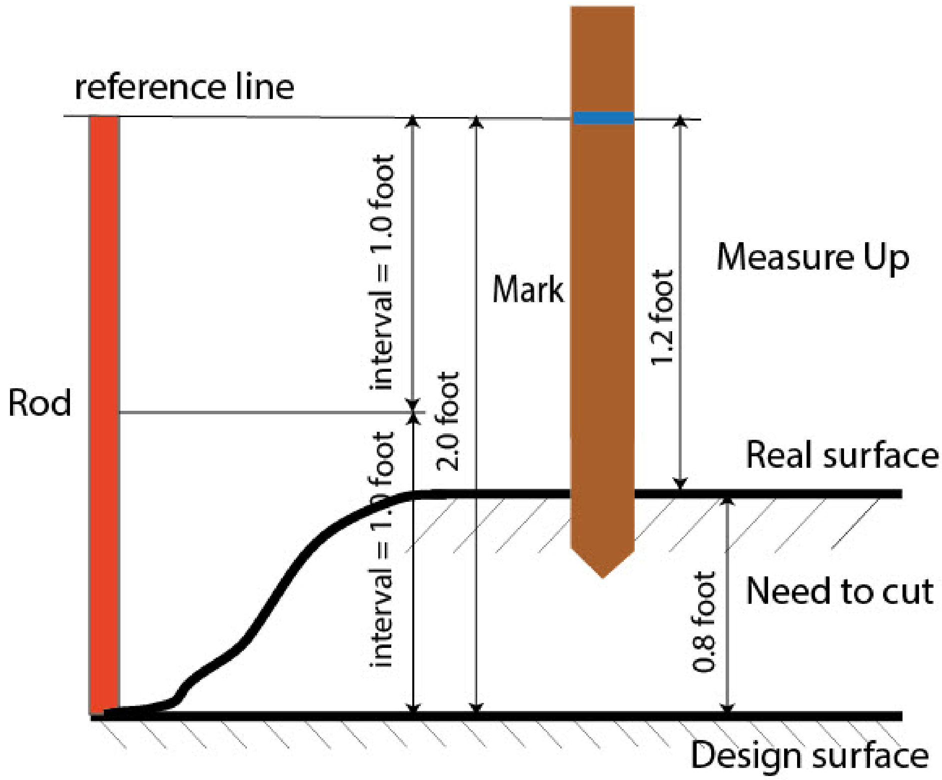
17. MAGNET Relay: Grade Stake Marking

This dialog allows you to mark the grade stake with the skyward rounding of the cut/fill decimal values to an even number of feet (or meters).

To configure the settings for marking:

1. In the **Working Stake Length** field, enter the working length of the stake.
2. In **Top Stake Spacing** field, enter a value of top reserve part of the stake for marking.
3. In **Bottom Stake Spacing** field, enter a value of bottom reserve part of the stake for marking.
4. In **Cut/Fill interval** field, enter an even number you want the cut/fill to round to.
5. If all requirements are met, clicking **Next** opens the next **Stake Settings** dialog. More..

For instance if the cut is 0.8 and the bottom spacing is 0.5, we will round up to a cut of 2.0 (assuming 1.0 interval) and place the mark at 1.2 above ground:



18. MAGNET Relay: Stake Settings

In the *Store Staked Point As* fields you can set the parameters for storing staked points:

1. In the **Point** field, select a method to set the name for the first staked point. The initial point name can be set to:
   - *Design point* name.
   - *Next point* name.
   - *Design point with a pre-defined prefix* (that is, stk_01, where "stk_" is the prefix).
   - *Design point with a pre-defined suffix*. The choice of the default prefix or suffix appears only when the corresponding item is chosen from the drop-down menu.
   - *Design Point plus Constant*. A specified numerical constant can be added to automatically generate the staked point name. For instance, if the constant specified is 1000, and the design point is 100, the staked point would be named 1100 (that is, 100+1000). If the design point is alphanumeric, the

constant is appended to the name. For example, for the design point ALPHA, the corresponding staked out point is named ALPHA1000.
   - *Range Start*. Any start value of a range can be selected.
2. The **Note** can be set to either *Design point*name, *Design point with a prefix*, or *Design point with a suffix*. Also, it can be *Station & Offset* information. If the *Station & Offset* option is activated, an edit box for entering an alphanumeric prefix appears. For the United States, this prefix is "*Sta*", for the international markets the prefix is "*Cha*", and for the Korean/Japanese markets the prefix is "*No*". With this option activated, depending on the choice for the prefix, MAGNET Field automatically generates one note for each stakeout point: Sta5+5.5R5.0, Cha505.5R5.0, or No.5+5.5R5.0 respectively.

19. MAGNET Relay: Tracking

Select one of the following checkboxes to configure the satellites tracking in the receiver:

- Select **Track BDS Satellites** — to track BeiDou satellites.
- Select **Track IOV Satellites** — to track Galileo satellites.
- Select **Track SBAS Satellites** — to track SBAS satellites.
- Select **Track QZSS Satellites** — to track QZSS satellites.

Select one of the following checkboxes to configure the signal tracking for satellites::

- Select **Track L5 Signal** — to track GPS L5 and QZSS L5 signals.
- Select **Track L2C Signal** — to track GPS L2C, GLONASS C/A L2, QZSS L2C and BeiDou B2 signals.
- Select **Track GLO P Signal** — to track GLONASS P-codes on L1 and L2.

From the *Satellite System* list you can select a satellite constellation to be used for position computation for Standalone, DGPS and RTK solutions:

- *ALL* :
  - GPS satellites with L1C/A, L1P and L2P signals, plus:
    - L5 signals, if the **Track L5 Signal** check box is selected;
    - L2C signals, if the **Track L2C Signal** check box is selected.
  - GLONASS satellites with L1C/A, L1P signals, plus:
    - L2C signals, if the **Track L2C Signal** check box is selected;
    - L2P signals, if the check box **Track GLO L2P Signal** is selected.
  - If the **Track BDS Satellites** checkbox is selected — BeiDou satellites will be used. By default, the B1 signals are available. If the **Track L2C Signal** checkbox is selected, the B2 signals are available.
  - If the **Track QZSS Satellites** checkbox is selected — QZSS satellites will be used. By default, the L1C/A signals are available. If the **Track L2C Signal** checkbox is selected, the L2C signals are available.
  - If the **Track SBAS Satellites** checkbox is selected — SBAS satellites will be used. By default, the L1C/A signals are available.

- *GPS* - GPS satellites with L1C/A, L1P and L2P signals, plus
  - L5 signals, if the **Track L5 Signal** check box is selected;
  - L2C signals, if the **Track L2C Signal** check box is selected.
- *GPS + GLONASS*:
  - GPS satellites with L1C/A, L1P and L2P signals, plus
    - L5 signals, if the **Track L5 Signal**check box is selected;
    - L2C signals, if the **Track L2C Signal** check box is selected;
  - GLONASS satellites with L1C/A, L1P signals, plus
    - L2C signals, if the **Track L2C Signal** check box is selected;
    - L2P signals, if the **Track GLO L2P Signal** check box is selected.
- *GPS + BDS*:
  - GPS satellites with L1C/A, L1P and L2P signals, plus
    - L5 signals, if the **Track L5 Signal**check box is selected;
    - L2C signals, if the **Track L2C Signal** check box is selected;
  - If the **Track BDS Satellites** checkbox is selected — BeiDou satellites will be used. By default, the B1 signals are available. If the **Track L2C Signal** checkbox is selected, the B2 signals are available.
- *GPS + GLONASS + BDS*:
  - GPS satellites with L1C/A, L1P and L2P signals, plus
    - L5 signals, if the **Track L5 Signal**check box is selected;
    - L2C signals, if the **Track L2C Signal** check box is selected;
  - GLONASS satellites with L1C/A, L1P signals, plus
    - L2C signals, if the **Track L2C Signal** check box is selected;
    - L2P signals, if the **Track GLO L2P Signal** check box is selected.
  - If the **Track BDS Satellites** checkbox is selected — BeiDou satellites will be used. By default, the B1 signals are available. If the **Track L2C Signal** checkbox is selected, the B2 signals are available.

Note: The selected configuration for tracking and positioning will work properly if you're using the hardware and firmware which supports the selected signals.

20. MAGNET Relay: Advanced

In this dialog you can select the signal processing parameters and RTK setting:

1. From **RTK Position**, select the method of RTK corrections definition:
   - *Extrapolation*(sometimes described as asynchronous) - the RTK engine will extrapolate the base station's carrier phase measurements to the current epoch (note that the truth carrier phases measured at the base cannot be transmitted and received at the rover instantly). The final positioning accuracy may be somewhat lower due to additional extrapolation errors, which may be up to a few millimeters vertical and horizontal for a one second extrapolation time.

     or

- *Matched Epoch* (sometimes described as synchronous) - the RTK engine does not extrapolate the base station's carrier phases in position computation. Instead, the engine will either compute a delayed position or simply output the current stand-alone position (while waiting for new RTCM/CMR messages from the base station). Note that the delayed position is computed for the time (epoch) to which the last received base station's carrier phase measurements correspond. Accuracies achievable in delay mode are normally on a level with those of post-processing kinematic.

2. By default the **Multipath Reduction** is selected to enable the use of a special signal processing technique for reduction of C/A code phase multipath and C/A carrier phase multipath. This option is useful for collecting raw data near from metallic objects, or trees and high buildings.
3. Select the **Canopy Environment** check box to allow the RTK engine to use less rigid thresholds when filtering out measurement outliers. This mode is recommended when working under tree canopy or in other cases of high multipath.
4. If the GNSS receiver is collecting raw data in high-vibration environment, we recommend to select **High-Vibration Environment (QLL)** to enable Quartz Lock Loop technology to minimize the vibration-induced impact on acquisition and tracking capabilities of the TPS receiver.
5. Optionally, you can select the settings of the RTK engine. To do this, select the RTK Settings option from the pop-up menu that displays by clicking  in the top left corner.
6. Click **Next**. The wizard will open the last screen to create the *MAGNET Relay* configuration.

21. MAGNET Relay: Miscellaneous

In this dialog you can you to customize the user interface:

1. **Display coordinates after measurement**: when selected, computed coordinates are displayed automatically after a GPS measurement is performed and before the point coordinates are stored into the database.
2. **Prompt for antenna height**: when selected, prompts for a height of the antenna before a point is stored.
3. **Beep on storing points**: by default this is turned on to beep each time the point is stored.
4. **Restart Epoch Counter if Solution changes**: when selected , counting epochs for will stop if the specified solution is lost and will resume after it is found.

Real Time DGPS/NMEA Survey

Real Time Differential GPS implies that the rover receiver uses differential pseudorange correction data transmitted from DGPS services. Real Time DGPS survey is used in GIS applications. A number of differential services exist to transmit differential correctional data, including maritime radio beacons, and SBAS (Satellite - Based Augmentation Systems) service.

1. Real Time DGPS/NMEA: Configuration

To add / edit a new configuration:

1. Enter the **Name** for the configuration that will be displayed in the Configurations list.
2. In the field **Type** select the *Real Time DGPS/NMEA* configuration.
3. In the **Corrections** field select the type of correction data that will be used for survey.
   - *User Base* - a user base transmits pseudo-range corrections. This configuration is similar to RTK with the pseudo-range measurements only.
   - *Beacon* - a radio beacon transmits pseudo-range corrections.
   - *SBAS/Autonomous* - the Satellite-Based Augmentation Systems (WAAS, EGNOS, or MSAS) are source of differential correction data. The availability of satellite signals depends upon the receiver type and location.
   - *CDGPS* - the Canada-wide Differential GPS (CDGPS) Service provides wide-area DGPS corrections via L-band communications satellite across the breadth of Canada as well as parts of the United States.
4. Optionally, you can set a value to increment/ decrement the survey point number when adding a new point.

   To do this, select the Point Properties option from the pop-up menu that displays by clicking  in the top left corner.
5. Click **Next**. The wizard will open the corresponding screens to create the *Real Time DGPS/NMEA* configuration.

2. Real Time DGPS/NMEA: Receiver Make

1. If you will work with real GNSS receivers, select the vendors of receivers. Do the following:
   - for the *Use Base* corrections: select the Base receiver and Rover receiver from the list in the **Base** and **Rover** fields.
   - for the *Beacon*, *SBAS/Autonomous* and *CDGPS* corrections: select the Rover receiver from the list in the **Rover** fields.
2. If you will work without receivers, select the **Simulation Mode** check box. You can set the simulation parameters in the Simulation Setup dialog.
3. Click **Next**. The wizard will open the corresponding screen to create the *Real Time DGPS/NMEA* configuration.

   Note: For *SBAS/Autonomous* corrections you can select *Generic NMEA* in the **Manufacturer** drop-down selection list. For this manufacturer you cannot set the **Simulation mode** for the rover receivers.

Beacon

This dialog is available for the *Beacon* corrections only.

To configure settings for a radio beacon source for differential GPS corrections:

1. Select the **Country** where the radio-beacon based differential service is located.
2. Select the **Station** that provides broadcasting differential corrections for the rover.

3. Select the **Beacon Corrections from BR-1** check box if required to use the beacon receiver BR-1 as a source of differential corrections for the rover. More...
4. Select the **Automatic Scan Mode** check box if you want to enable this mode in BR-1 to get the Beacon signal automatically. BR-1 will search broadcasting frequencies and output RTCM corrections from the best signal.

3. Real Time DGPS/NMEA: Base Receiver

This dialog opens for *User Base* corrections only. To configure the base receiver:

1. The **Ext. Receiver** box is enabled for a controller that has an internal GNSS receiver. You can select a connection with the internal GNSS receiver or any external GNSS receiver. If the controller does not have a GNSS receiver or the software is installed on the computer, the **Ext. Receiver** box is always checked. The connection for the external receivers can be *Bluetooth* and *Serial Cable*.
2. From the **Receiver Model** list, select the model of the base receiver you are using and enter its **Serial Number**.
3. Set **Elevation Mask**. Usually the default 13 degrees is appropriate. Data from satellites below this elevation will not be used.
4. From **DGPS Format** list, select the format for the differential corrections data, which the base radio will transmit.
5. From the **Antenna** drop-down list, select the type of the base receiver antenna used.
6. Enter the antenna height and set the type of height measurement (vertical or slant). To do this, click the **Antenna Height** button and select *Edit* from the drop-down menu. The Antenna Setup dialog is opened. Here you can edit the antenna type, the value of the antenna height, and the type of height.
7. If needed to transmit data from different ports of the base receiver to several rovers, you need activate the **Multiple Ports** option. To do this, click the **Peripherals** button. The Peripherals dialog is opened. Here you can select the number of ports to use for base output.
8. Optionally, you can turn off the charger mode for the receiver's internal battery if it is available. To do this, select the Receiver Settings option from the pop-up menu that displays by clicking  in the top left corner.
9. Click **Next**. The wizard will open the corresponding screen to create the *Real Time DGPS/NMEA* configuration.

5. Real Time DGPS/NMEA: Base Radio

The base radio is intended for transmitting differential corrections. You can set GSM/CDMA modem (*Cellular* modem) or UHF modem (*Radio*) or *LongLINK* for HiPer SR receiver. The dialog content depends on the receiver model selected.

To configure the radio modem:

1. In the **Device Type** field select which type of Radio Modem you will use: *Internal Radio*, or *Internal Cellular*, or *External Radio*, or *External Cellular* (for Topcon Generic, HiPer II, HiPer V, GR-3,GR-5, NET G3).
2. In the **Device Model** field select model of the modem (for Topcon Generic, HiPer SR,HiPer II, HiPer V, GR-3,GR-5, NET G3). For HiPer SR receivers, the *Long LINK* modem does not require any additional settings.
3. For *Internal Radio*, *External Radio*, *External Cellular* specify the **Baud** rate for the receiver **Port** that the radio is connected to.

   Note: Sometimes during modem connection, for the "*Auto*" baud rate selection of the modem, the baud rate could not be set. The software creates an error message for this case: *" Modem baud rate could not be determined automatically. Please run TRU (see Help for details) or specify baud rate in configuration*". See here how to resolve the issue.

4. For *External Radio*, *External Cellular* specify communication parameters for the receiver **Port** that the radio is connected to: **Parity**, the number of **Data** bits, the number of **Stop** bits, which are specific to the connected modem.
5. If you use an AirLink CDMA or GPRS external cellular modem, select the *Modem Register* option from the pop-up menu that displays by clicking  in the top left corner to register the radio modem. More...
6. If required, click **Defaults** to return all the communication parameters to the default settings.
7. Click **Next** to specify the base and rover modem radio parameters as required.

Real Time DGPS/NMEA: Base and Rover Radio Parameters

The type of the dialog is dependent upon the selected modem type for the base or radio receiver. Click the desired modem type to open an instruction how to configure modem parameters:

1. Internal Radio:
   - Digital UHF / Digital UHFII
   - FH 915 Plus
   - Satel
2. Internal Cellular
   - Auto, Digital UHF I/II GSM,FH915 + GSM, General Internal GSM, Satel GSM,
   - Digital UHF CDMA
   - TCP/IP
3. External Radio
   - RE-S1
   - Satel, SRL-35
   - TR-35
4. External Cellular

- AirLink CDMA (MUDP) for base
- AirLink GPRS for rover
- AirLink CDMA for rover
- Generic CDMA for rover
- Generic GSM, MultiTech GSM, Siemens TC35
- TCP/IP for base

You can set GSM/CDMA modem (*Cellular* modem) or UHF modem (*Radio*). The dialog content depends on the receiver model selected.

6. Real Time DGPS/NMEA: Rover Receiver

To configure the rover receiver:

1. The **Ext. Receiver** box is enabled for a controller that has an internal GNSS receiver. You can select a connection with the internal GNSS receiver or any external GNSS receiver. If the controller does not have a GNSS receiver or the software is installed on the computer, the **Ext. Receiver** box is always checked. The connection for the external receivers can be *Bluetooth* and *Serial Cable*. Note: *Topcon Generic* selection will work with all receiver models which were manufactured before GR-3 receiver.
2. From the **Receiver Model** list, select the model of the rover receiver you are using and enter its **Serial Number**.
3. Set **Elevation Mask**. Usually the default 13 degrees is appropriate. Data from satellites below this elevation will not be used.
4. From **DGPS Format** list, select the format for the differential corrections data, which the rover radio modem will receive (For the *Generic NMEA* this field is hidden).
5. From the **Antenna** drop-down list, select the type of the rover receiver antenna used.
6. Enter the antenna height and set the type of height measurement (vertical or slant). To do this, click the **Antenna Height** button and select *Edit* from the drop-down menu. The Antenna Setup dialog is opened. Here you can edit the antenna type, the value of the antenna height, and the type of height.
7. If needed to transmit data from different ports of the base receiver to several rovers, you need activate the **Multiple Ports** option. To do this, click the **Peripherals** button. The Peripherals dialog is opened. Here you can select the number of ports to use for base output.
8. Optionally, you can turn off the charger mode for the receiver's internal battery if it is available. To do this, select the Receiver Settings option from the pop-up menu that displays by clicking  in the top left corner.
9. Click **Next**. The wizard will open the corresponding screen to create the *Real Time DGPS/NMEA* configuration.

7. Real Time DGPS/NMEA: Rover Modem

To configure the modem connection (for *User Base* corrections only):

1. Select either *Receiver*, or *Controller* depending on whose modem will use for communication.
2. Click **Next**. The wizard will open the corresponding screen to create the *Real Time DGPS/NMEA* configuration.

8. Real Time DGPS/NMEA: Rover Radio

The rover radio is intended for receiving differential corrections. The content of the dialog depends on the selected corrections type .

For the User Base corrections

You can set GSM/CDMA modem (*Cellular* modem) or UHF modem (*Radio*). The dialog content depends on the receiver model selected.

To configure the radio modem:

1. In the **Device Type** field select which type of Radio Modem you will use: *Internal Radio*, or *Internal Cellular*, or *External Radio*, or *External Cellular* (for Topcon Generic, HiPer II, HiPer V, GR-3,GR-5, NET G3).
2. In the **Device Model** field select model of the modem (for Topcon Generic, HiPer SR,HiPer II, HiPer V, GR-3,GR-5, NET G3). For HiPer SR receivers, the *Long LINK* modem does not require any additional settings.
3. For *Internal Radio*, *External Radio*, *External Cellular* specify the **Baud** rate for the **Port** to which the radio is connected.]

   Note: Sometimes during modem connection, for the "*Auto*" baud rate selection of the modem, the baud rate could not be set. The software creates an error message for this case: *" Modem baud rate could not be determined automatically. Please run TRU (see Help for details) or specify baud rate in configuration*". See here how to resolve the issue.

4. For *External Radio*, *External Cellular* specify communication parameters for the receiver **Port** that the radio is connected to: **Parity**, the number of **Data** bits, the number of **Stop** bits, which are specific to the connected modem.
5. If you use an AirLink CDMA or GPRS external cellular modem, select the *Modem Register* option from the pop-up menu that displays by clicking  in the top left corner to register the radio modem. More...
6. If required, click **Defaults** to return all the communication parameters to the default settings.
7. Click **Next** to specify the base and rover modem radio parameters as required.

For the Beacon corrections

If the **Beacon Corrections from BR-1** check box is selected (in the Config: Beacon dialog), this dialog appears.

To configure the BR-1 connection:

1. Select either *Receiver*, or *Controller* depending on whose BR-1 will use for communication.
2. In the **Port Connected to BR-1** panel specify the **Baud** rate for the **Port** to which the BR-1 is connected (*Bluetooth* or *COM1*).
3. If required, click **Defaults** to return all the communication parameters to the default settings.
4. Click **Next**. The wizard will open the corresponding screen to create the *Real Time DGPS/NMEA* configuration.

SBAS Setup

This dialog opens for *SBAS/Autonomous* corrections.

To configure SBAS:

1. For all new Topcon receivers that support automatic tracking, you can select one of two options for use of SBAS satellites: **Best Available** or **Custom**.
   Note : If the **Topcon Generic** receiver was selected in the *Rover Receiver* dialog, custom setup is required. Not more than two SBAS satellites can be enabled for tracking in Topcon Generic receiver.
2. For custom selection, select the boxes near the PRN numbers of the satellites as required.
   Note: All satellites can be selected. The satellite most available from those selected will be used in DGPS solution.
   - PRN #, Name and Type: These columns will list all possible SBAS PRN numbers with the respective names of the satellites and types of satellite systems.
   - GPS #: This PRN # applies only to the Topcon Generic receiver. One of currently unused GPS numbers should be selected in this column to be able to track this satellite in Satellite View dialog. To change, click GPS number and select the appropriate number from the pop-up menu.
3. Enable/disable use of ionospheric corrections from the SBAS satellite when computing positions. It is recommended to use ionospheric corrections.
   - *None*: ionospheric corrections are not used
   - *Apply if avail*: use ionospheric corrections if available
   - *Use sat only if avail*: use only the satellites for which ionospheric corrections are available.

11. Real Time DGPS/NMEA: Topo Survey

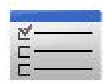
During a real-time stationary survey you can select two independent ways to save measured points to the current job: **Precise** and **Quick**. In the Topo dialog or Stake dialog, the **Precise** way will be activated by clicking , and the **Quick** way will be activated by clicking .

In the **Precise** area you can set the following parameters:

1. **Solution**: for the *Real Time DGPS/NMEA* configuration *All* and *DGPS* types solution will be considered. More...
2. **Measure Continuously**: select the check box to log measurements continuously and stop them manually when required.
3. **Average**: the field allows you to set a specific number of measurements for logging and average them for storing the position. If needed, you can change the default number (3) of measurements.
4. **Precision**: select the check box to consider the threshold of the horizontal and vertical precisions of measurement to store the position. The default values are 0.25 m and 0.400 m, respectively.
5. **Auto Store** : select the check box to enable auto-storing positions for the average mode.

In the **Quick** area you can set the following parameters:

1. **Solution**: for the *Real Time DGPS/NMEA* configuration *All* and *DGPS* types solution will be considered. More...
2. **Average**: the field allows you to set a specific number of measurements for logging and average them for storing the position. If required, you can change the default number (1) of measurements.
3. **Precision**: select the check box to consider the threshold of the horizontal and vertical precisions of measurement to store the position. The default values are 0.25 m and 0.400 m, respectively.

Note: Every survey parameter can be changed with the help of the  button from any in the Topo and Stake dialog in GPS+ mode.

Click **Next**. The wizard will open the corresponding screen to create the *Real Time DGPS/NMEA* configuration.

12. Real Time DGPS/NMEA: Auto Topo Survey

For for auto-storing positions in the dynamic Real Time DGPS/NMEA survey:

1. **Solution**: from the drop-down list, select the solution type for each position computation as required. More...
2. **Method** : from the drop-down list, select the method for setting the interval between the received epochs: by time, by horizontal distance, by slope distance or by elevation.
3. **Interval**: in the field, enter the value for the selected method:
   - in seconds for the *By Time* method. Default value is 1 second.
   - in meters for the *By Horiz Dist*, *By Slope Dist* and *By Elevation*. Default value is 1 meter.
4. Click **Next**. The wizard will open the corresponding screen to create the *Real Time DGPS/NMEA* configuration.

13. Real Time DGPS/NMEA: Stake Settings

To configure the parameters which will be used during a GPS stakeout:

1. In the **Horizontal Distance Tolerance** filed, enter the horizontal distance tolerance value. It is used to determine when you are close enough to the point for the bull's eye to show up.
2. In the **Screen Orientation**field you can select orientation of the screen during stakeout:
   - *North* - the top of the screen is oriented to the north during the stakeout session;
   - *Moving Direction* - the top of the screen is oriented to the direction of moving during the stakeout session;
   - *Moving Direction +North* - is similar to the *Moving Direction* option, but the screen will be oriented to the *North* direction when you are within three meters of the design point;
   - *Point/Azimuth*- if you select this orientation type, the *Orient Pt / Orient Az* field will be added to the **Stake Point** dialog. Here you can define a point or an azimuth to orient the screen top.
3. From the **Display Reference** drop-down list, select a object that will be displayed used in the *Normal View* mode.
4. Optionally, you can specify an icon for the staked point. To do this, select the Display option from the pop-up menu that displays by clicking  in the top left corner.
5. Click **Next**. The wizard will open the corresponding screen to create the *Real Time DGPS/NMEA* configuration.

14. Real Time DGPS/NMEA: Stake Survey

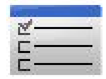
During a stakeout procedure you can select two independent ways to save stake out points to the current job: **Precise** and **Quick**. In the Topo dialog or Stake dialog, the **Precise** way will be activated by clicking  , and the **Quick** way will be activated by clicking .

In the **Precise** area you can set the following parameters:

1. **Solution**: for the *Real Time DGPS/NMEA* configuration *All* and *DGPS* types solution will be considered. More...
2. **Measure Continuously**: select the check box to log measurements continuously and stop them manually when required.
3. **Average**: the field allows you to set a specific number of measurements for logging and average them for storing the position. If needed, you can change the default number (3) of measurements.
4. **Precision**: select the check box to consider the threshold of the horizontal and vertical precisions of measurement to store the position. The default values are 0.25 m and 0.400 m, respectively.
5. **Auto Store** : select the check box to enable auto-storing positions for the average mode.

In the **Quick** area you can set the following parameters:

1. **Solution**: for the *Real Time DGPS/NMEA* configuration *All* and *DGPS* types solution will be considered. More...
2. **Average**: the field allows you to set a specific number of measurements for logging and average them for storing the position. If required, you can change the default number (1) of measurements.
3. **Precision**: select the check box to consider the threshold of the horizontal and vertical precisions of measurement to store the position. The default values are 0.25 m and 0.400 m, respectively.

Note: Every survey parameter can be changed with the help of the  button from any in the Topo and Stake dialog in GPS+ mode.

Click **Next**. The wizard will open the corresponding screen to create the *Real Time DGPS/NMEA* configuration.

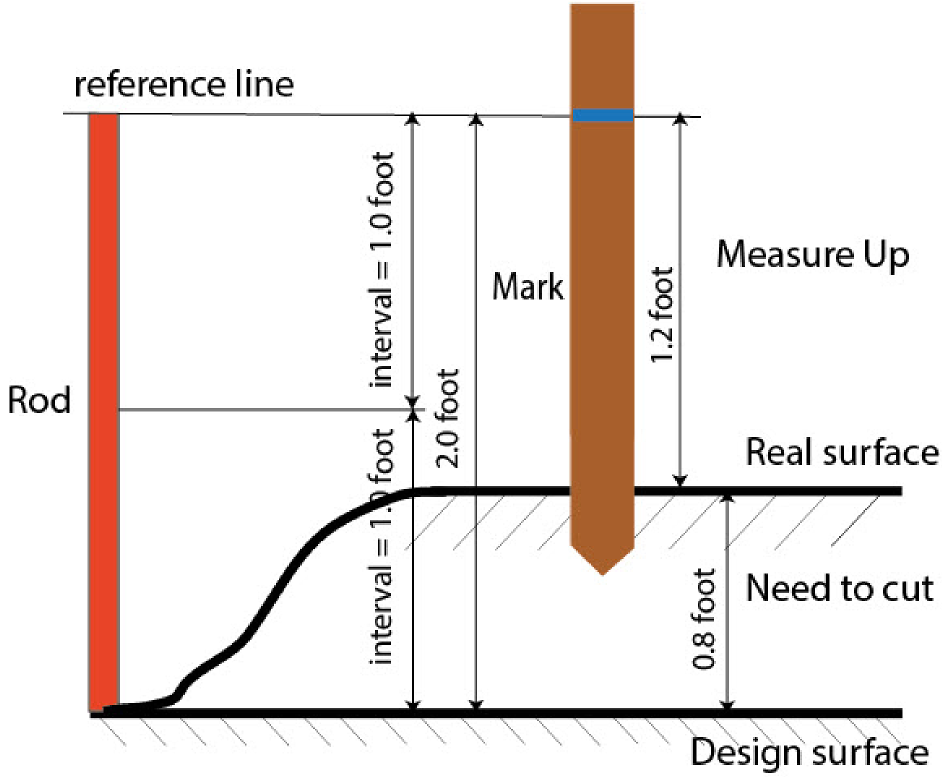
15. Real Time DGPS/NMEA: Grade Stake Marking

This dialog allows you to mark the grade stake with the skyward rounding of the cut/fill decimal values to an even number of feet (or meters).

To configure the settings for marking:

1. In the **Working Stake Length** field, enter the working length of the stake.
2. In **Top Stake Spacing** field, enter a value of top reserve part of the stake for marking.
3. In **Bottom Stake Spacing** field, enter a value of bottom reserve part of the stake for marking.
4. In **Cut/Fill interval** field, enter an even number you want the cut/fill to round to.
5. If all requirements are met, clicking **Next** opens the next **Stake Settings** dialog. More..

For instance if the cut is 0.8 and the bottom spacing is 0.5, we will round up to a cut of 2.0 (assuming 1.0 interval) and place the mark at 1.2 above ground:



16. Real Time DGPS/NMEA: Stake Settings

In the *Store Staked Point As* fields you can set the parameters for storing staked points:

1. In the **Point** field, select a method to set the name for the first staked point. The initial point name can be set to:
   - *Design point* name.
   - *Next point* name.
   - *Design point with a pre-defined prefix* (that is, stk_01, where "stk_" is the prefix).
   - *Design point with a pre-defined suffix*. The choice of the default prefix or suffix appears only when the corresponding item is chosen from the drop-down menu.
   - *Design Point plus Constant*. A specified numerical constant can be added to automatically generate the staked point name. For instance, if the constant specified is 1000, and the design point is 100, the staked point would be named 1100 (that is, 100+1000). If the design point is alphanumeric, the constant is appended to the name. For example, for the design point ALPHA, the corresponding staked out point is named ALPHA1000.
   - *Range Start*. Any start value of a range can be selected.
2. The **Note** can be set to either *Design point*name, *Design point with a prefix*, or *Design point with a suffix*. Also, it can be *Station & Offset* information. If the *Station & Offset* option is activated, an edit box for entering an alphanumeric prefix appears. For the United States, this prefix is "*Sta*", for the international markets the prefix is "*Cha*", and for the Korean/Japanese markets the prefix is "*No*". With this option activated,

depending on the choice for the prefix, MAGNET Field automatically generates one note for each stakeout point: Sta5+5.5R5.0, Cha505.5R5.0, or No.5+5.5R5.0 respectively.

17. Real Time DGPS/NMEA: Tracking

Select one of the following checkboxes to configure the satellites tracking in the receiver:

- Select **Track BDS Satellites** — to track BeiDou satellites.
- Select **Track IOV Satellites** — to track Galileo satellites.
- Select **Track SBAS Satellites** — to track SBAS satellites.
- Select **Track QZSS Satellites** — to track QZSS satellites.

Note: If the *SBAS/Autonomous* correction is selected, the **Track SBAS Satellites** is always checked.

Select one of the following checkboxes to configure the signal tracking for satellites::

- Select **Track L5 Signal** — to track GPS L5 and QZSS L5 signals.
- Select **Track L2C Signal** — to track GPS L2C, GLONASS C/A L2, QZSS L2C and BeiDou B2 signals.
- Select **Track GLO P Signal** — to track GLONASS P-codes on L1 and L2.

From the *Satellite System* list you can select a satellite constellation to be used for position computation for Standalone, DGPS and RTK solutions:

- *ALL* :
  - GPS satellites with L1C/A, L1P and L2P signals, plus:
    - L5 signals, if the **Track L5 Signal** check box is selected;
    - L2C signals, if the **Track L2C Signal** check box is selected.
  - GLONASS satellites with L1C/A, L1P signals, plus:
    - L2C signals, if the **Track L2C Signal** check box is selected;
    - L2P signals, if the check box **Track GLO L2P Signal** is selected.
  - If the **Track BDS Satellites** checkbox is selected — BeiDou satellites will be used. By default, the B1 signals are available. If the **Track L2C Signal** checkbox is selected, the B2 signals are available.
  - If the **Track QZSS Satellites** checkbox is selected — QZSS satellites will be used. By default, the L1C/A signals are available. If the **Track L2C Signal** checkbox is selected, the L2C signals are available.
  - If the **Track SBAS Satellites** checkbox is selected — SBAS satellites will be used. By default, the L1C/A signals are available.
- *GPS* - GPS satellites with L1C/A, L1P and L2P signals, plus
  - L5 signals, if the **Track L5 Signal** check box is selected;
  - L2C signals, if the **Track L2C Signal** check box is selected.
- *GPS + GLONASS*:

- GPS satellites with L1C/A, L1P and L2P signals, plus
    - L5 signals, if the **Track L5 Signal**check box is selected;
    - L2C signals, if the **Track L2C Signal** check box is selected;
- GLONASS satellites with L1C/A, L1P signals, plus
    - L2C signals, if the **Track L2C Signal** check box is selected;
    - L2P signals, if the **Track GLO L2P Signal** check box is selected.
- *GPS + BDS*:
  - GPS satellites with L1C/A, L1P and L2P signals, plus
    - L5 signals, if the **Track L5 Signal**check box is selected;
    - L2C signals, if the **Track L2C Signal** check box is selected;
  - If the **Track BDS Satellites** checkbox is selected — BeiDou satellites will be used. By default, the B1 signals are available. If the **Track L2C Signal** checkbox is selected, the B2 signals are available.
- *GPS + GLONASS + BDS*:
  - GPS satellites with L1C/A, L1P and L2P signals, plus
    - L5 signals, if the **Track L5 Signal**check box is selected;
    - L2C signals, if the **Track L2C Signal** check box is selected;
  - GLONASS satellites with L1C/A, L1P signals, plus
    - L2C signals, if the **Track L2C Signal** check box is selected;
    - L2P signals, if the **Track GLO L2P Signal** check box is selected.
  - If the **Track BDS Satellites** checkbox is selected — BeiDou satellites will be used. By default, the B1 signals are available. If the **Track L2C Signal** checkbox is selected, the B2 signals are available.

Note: The selected configuration for tracking and positioning will work properly if you're using the hardware and firmware which supports the selected signals.

18. Real Time DGPS/NMEA: Advanced

In this dialog you can select the signal processing parameters:

1. By default the **Multipath Reduction** is selected to enable the use of a special signal processing technique for reduction of C/A code phase multipath and C/A carrier phase multipath. This option is useful for collecting raw data near from metallic objects, or trees and high buildings.
2. If the GNSS receiver is collecting raw data in high-vibration environment, we recommend to select **High-Vibration Environment (QLL)** to enable Quartz Lock Loop technology to minimize the vibration-induced impact on acquisition and tracking capabilities of the TPS receiver.
3. Click **Next**. The wizard will open the last screen to create the *Real Time DGPS/NMEA* configuration.

19. Real Time DGPS/NMEA: Miscellaneous

In this dialog you can you to customize the user interface:

1. **Display coordinates after measurement**: when selected, computed coordinates are displayed automatically after a GPS measurement is performed and before the point coordinates are stored into the database.

2. **Prompt for antenna height**: when selected, prompts for a height of the antenna before a point is stored.
3. **Beep on storing points**: by default this is turned on to beep each time the point is stored.
4. **Restart Epoch Counter if Solution changes**: when selected , counting epochs for will stop if the specified solution is lost and will resume after it is found.

Other settings

In this part you can find the description of configuration of external devices and modems.

Receiver Settings

When an external power supply is used at the base, and the receiver charger mode is turned on, it supplies power to the receiver battery. You can use an external power supply without charging the receiver battery.

To do this:

1. Select the **Turn Charger Mode Off** check box.
2. Click  to save the setting.

Point Properties

Set a value to increment/ decrement the survey point number when adding a new point.

Tracking

In this dialog you can configure the GNSS receiver to track the satellite systems and satellite signals and to select a satellite constellation to be used for position computation for Standalone, DGPS and RTK solutions for the following GPS+ survey types:

- Network RTK
- RTK
- MAGNET Relay
- Real Time DGPS/NMEA

RTK Settings

This dialog allows you to set the following parameters to configure the RTK engine.

In the **Ambiguity** box you can select the initial confidence level to the fixed RTK solution and defines the time period at which the ambiguity resolution is performed:

- **Level**: The RTK engine has 3 levels of confidence when fixing integer ambiguities, *Low*, *Medium* and *High* which correspond to the indicator's 95%, 99.5% and 99.9% thresholds, respectively. The receiver's RTK engine will constantly update the confidence level indicator as new measurements arrive. Once this parameter exceeds the selected threshold, the engine will fix up all or some of the integer ambiguities. The corresponding position estimate will be marked as fixed RTK solution. The higher the confidence level specified, the longer the integer ambiguity search time. By default, the *Medium* is set.

- **Resolution Period**: defines the interval in seconds at which the ambiguity resolution is performed. This interval is used for *Extrapolation* and *Matched Epoch* modes.

- **Correction Period** sets:
  - the differential interval for the base station, when you configure a Base receiver
  - the same interval to output the RTK position (by the rover receiver), when you configure a Rover receiver.

  By default, 1 second is set. Any updating of the value is only effective in the *Matched Epoch* mode.

- **Accept noisy observables** - After clicking the checkbox, the RTK engine will use less rigid thresholds for computing RTK solution. This parameter allows the user to speed up the ambiguity fixing procedure, but it also decreases the reliability of the ambiguity resolution. This parameter can be used for abnormal survey conditions only:
  - When a rover receiver gets VRS corrections from a network, we recommend to select the mode for a poor VRS network geometry when the receiver is located at the boundary of the VRS coverage area and the nearest physical station is located at a great distance.
  - When a rover receiver gets the correction from another receiver, we recommend to select the mode when the rover receiver is located close to metallic objects, high building and trees.
- **Set IF Baseline** - After clicking the checkbox, the RTK engine will use the ionosphere-free combination when the baseline length is greater or equal to the entered value.
- **Set PDOP Mask** - After clicking the checkbox, the rover receiver will not compute the RTK position if the computed PDOP exceeds the specified threshold value.

Solution Types

There can be a combination of the following solution types:

- *Fixed mmGPS+*: positions were Fixed Only solution with mmGPS+ calculated height.

- *Fixed Only*: positions were computed by RTK engine using the carrier phase measurements from Base and Rover receivers. Integer ambiguities were fixed.
- *Float mmGPS+*: positions were Float solution with mmGPS+ calculated height.
- *Float*: positions were computed by RTK engine using the carrier phase measurements from base and rover receivers. Integer ambiguities, however, were NOT fixed (their float estimates were used instead).
- *DGPS*: positions were achieved using the pseudo-range measurements from base and rover receivers.

- *All*: positions were computed using all the epochs accepted, including autonomous solutions.
- *Auto*: autonomous positions were computed when differential corrections are not available.

Peripherals for the base receiver

Select the **Multiple Ports** check box, and then select the number of ports to use for base output if required.

Additional dialogs will appear later in the wizard sequence to setup parameters for each port.

Peripherals for the rover receiver

In this dialog you can select any option to work with peripherals of the rover receiver:

1. Select the **NMEA Ports** check box to configure the output of the NMEA messages. Select the number of ports from the drop-down list that appears and click . Additional dialogs will appear later in the wizard sequence to setup parameters for each port and specify NMEA messages for output. More...
2. Select the **Multiple Ports** check box and then select the number of ports to use for rover output if required. Additional dialogs will appear later in the wizard sequence to setup parameters for each port. This check box is available for *RTK* type. Note: Use only one port (one radio modem) to receive corrections from the base.
3. Select the **Depth Sounder** check box to configure a depth sounder. Click **Parameters** to specify settings. More...
4. Select the **mmGPS+** check box to configure the mmGPS+ system. Click **Parameters** to specify settings. This check box is available for *RTK* and *Network RTK* types. More...
5. Select the **External Laser** check box to configure the laser device that can be connected either directly to the Controller or through the Receiver. Click **Parameters** to specify settings. More...
6. Select the **Use Repeater** check box to configure the repeater that can be connected either directly to the Controller or through the Receiver. Click **Parameters** to specify settings. This check box is available for *RTK* type. More...

Output NMEA

To configure the *NMEA Messages* for output:

1. Select the check boxes near the types of messages as required. More...
2. Select the **Set GP as Receiver Talker ID** check box to instruct the receiver to use "GP" as Talker ID in appropriate NMEA sentences generated. This enables support of Google Maps that cannot recognize default "GN" or "GL" as Talker IDs in these messages.
3. Enter the **Interval** in seconds in which the application will output the messages (up to 0.1 sec.).

List of NMEA messages

The following *NMEA Messages* are available for output:

- *GSA* to output the operation mode of the GNSS receiver, the satellite used for positioning, and DOP.
- *GLL* to output data on the current latitude/longitude and positioning mode.
- *VTG* to output the traveling direction and velocity.
- *GRS* to output the residual error of distance for each satellite. This is used to support RAIM.
- *ZDA* to output the UTC, day, month, year, and local time zone.
- *GST* to output the statistics of position errors.
- *GNS* to output data on time, position, and positioning of GPS+GLONASS (GNSS).
- *GGA* to output data on time, position, and positioning.
- *GSV* to output the number of satellites, satellite number, elevation angle, azimuthal angle, and SNR.
- *HDT* to output the direction (heading).
- *P_ATT* to output attitude parameters.
- *RMC* to output time, date, position, course and speed data provided by a GNSS navigation receiver.
- *ROT* to output rate of turn.
- *GMP* to output GNSS map projection fix data.

Configure Depth Sounder

If required, select **Simulation Mode** to test and demo depth sounder functionality without actually having a depth sounder in water.

To configure the *Depth Sounder*:

1. Select the **Model** of the depth sounder.
2. Set the **Max Depth Age** in second. This value determinate the age of the depth measurement which will be used in the future positioning calculation together with GPS measurements.
3. Set **Depth Sounder Port Setting** including port, parity, data, baud and stop rates to connect to the device.

When the configuration is used, the depth sounder icon( ) will appear on the status bar of any measurement dialog.

mmGPS+ Parameters

In mmGPS aided RTK survey, a wireless sensor connected to the Rover picks up the signals from the laser transmitter for accurate (millimeter) elevations.

Note: When measuring the height of the Rover antenna, include the height of the sensor with a 5/8 inch plug.

To configure the mmGPS+:

1. Select the **Receiver port***,* which is connected to the mmGPS+ Sensor.
2. Select the **Sensor Gain** to adjust the gain on the mmGPS+ Sensor. Select Auto to automatically control the mmGPS receiver's detection level of the transmitter's signal.
3. Enter the **Height Difference Limit** value to set the threshold for the difference between GPS and mmGPS+ height measurements. If the GPS+ height and mmGPS+ height differ by more than the amount entered, the mmGPS+ icon will change to warn the user.

Laser Configuration

In this dialog you can configure the external laser:

1. Select the laser **Manufacturer**. Currently MAGNET Field supports MDL, Leica and Laser Technology, Inc.
2. For the Laser Technology you can select the **Model** of the instrument.
3. Select the **Type** of laser measurement system if it uses Encoding or not.
4. If required, enter the offset value for the laser in the **Mount Offset** field.
5. In the **Device** field select *Receiver* if the external laser is connected with the receiver, or select *Controller* if the external laser is connected with the controller.
6. Specify communication parameters for the receiver **Port** that the laser is connected to: **Baud** rate, **Parity**, the number of **Data** bits, the number of **Stop** bits.

Repeater Configuration

If required, the you can use an external radio modem or receiver with an internal radio modem as a stand-alone repeater to increase the range between the Base and Rover.

To configure the repeater:

1. In the **Model** field, select model of the modem: either *Digital UHF*, or *Digital UHF II*, or *RE-S1*, or *Satel*.
2. In the **Connect Type** field, select *Receiver* if a modem is connected with the receiver, or select *Controller* if the external modem is connected with the controller.
3. In the **Connect Port Setup** field, specify communication parameters for the receiver/controller **Port** that the repeater is connected to: **Baud** rate, **Parity**, the number of **Data** bits, the number of **Stop** bits.
   Note: If you have selected *Receiver*and *Port C*, the internal modem of the *GNSS* receiver will be used as a repeater, and you cannot use the receiver for survey.
4. Click **Next** to continue the repeater configuration:
   - For *Digital UHF*, *Digital UHF II* see Radio Param.
   - For *RE-S1* see Radio Param.
   - For *Satel* see Radio Param.

Radio Parameters for Digital UHF / Digital UHF II and TRL-35

In this dialog you can configure internal radio modem *Digital UHF* and *Digital UHF II*, external modem TRL-35. Also you can configure modem *Digital UHF* and *Digital UHF II* as repeater:

1. Select the output **Power** value for the radio modem.
2. Select an appropriate operation **Protocol** for data transmitting/receiving:
   - *Simplex* protocol is ArWest's proprietary protocol. This protocol is used for Digital UHF and TRL-35 radio modem.

- *PDL* protocol can selected for any modem types.
- *TrimTalk* protocol is Trimble protocol and it can be used for any modem types.
- *Satel FCS Off* protocol is Satel protocol without Free Channel Scan (FCS). The protocol can be used for Digital UHF II only.
- *Satel FCS On* protocol is Satel protocol with Free Channel Scan (FCS). The protocol can be used for Digital UHF II only.

3. Select the type of **Modulation** for the radio modem. The modulation type is defined by the modem type and the selected protocol :
    - you can select between *DBPSK* and *DQPSK* modulation for Digital UHF and TRL-35 radio modems with *Simplex* protocol.
    - you can select *GMSK* modulation for Digital UHF and TRL-35 radio modems with *PDL* and *TrimTalk* protocol.
    - you can select between *GMSK* and *4-level FSK* for Digital UHF II radio modem with *PDL* protocol.
    - you can select *GMSK* for Digital UHF II and TRL-35 radio modems with *TrimTalk* protocol.
    - you can select *4FSK* for Digital UHF II radio modem with *Satel FCS Off*/ *Satel FCS On* protocol.
4. In the **Spacing** field you can select:
    - *12.5 kHz*- to provide more jam-resistant communication with low baud rate (4800 bps for *GMSK* and 9600 bps for *4-level FSK*)
    - *25 kHz* - to provide communication with high baud rate (9600 bps for *GMSK* and 19200 bps for *4-level FSK*)
    - *Do not set*- to save the previous setting of the Spacing parameter.

    The parameter depends on the chosen protocol, modulation type and link rate.

5. In the **Scrambling** field you can select:
    - *On* or (1-255) to activate Scrambling option — more robust data communication over high interference areas.
    - *Off* to deactivate the Scrambling option.
    - *Don't set up* to save the previous setting of the Scrambling option.

Scrambling option depends on the selected protocol and modem type:

- for the *Simplex* protocol you can activate, or deactivate, or stay previous Scrambling setting for Digital UHF and TRL-35 radio modems. To activate the option you need select a seed for pseudorandom sequence generator (the value from 1 to 255).
- for the *PDL* protocol you can activate, or deactivate, or stay previous Scrambling setting for Digital UHF, Digital UHF II and TRL-35 radio modems.
- for the *Trim Talk* protocol this option is:
    - automatically set for Digital UHF II radio modem.
    - saved previous Scrambling setting for Digital UHF and TRL-35 radio modems .
- for the *Satel* protocol this option is automatically switched on for Digital UHF II radio modem.

6. In the **FEC** field (Forward Error Correction) you can select:
   - *On* to activate the technique to control errors in data transmission over unreliable or noisy communication channels.
   - *Off* to deactivate the FEC option.
   - *Don't set up* to save the previous setting of the FEC option.

   FEC option depends on the selected protocol and modem type:

   - for the *Simplex* protocol you can activate, or deactivate, or stay previous FEC setting for Digital UHF and TRL-35 radio modems.
   - for the *PDL* protocol you can activate, or deactivate, or stay previous FEC setting for Digital UHF, Digital UHF II radio and TRL-35 modems.
   - for the *Trim Talk* protocol this option is:
     - automatically switched off for Digital UHF II radio modem.
     - saved previous FEC setting for Digital UHF and TRL-35 radio modems.
   - for the *Satel* protocol you can activate, or deactivate, or stay previous FEC setting for Digital UHF II radio modem.

Parameters for FH 915 Plus and RE-S1

In this dialog you can configure *FH 915 Plus* and *RE -S1* radio modem. Also you can configure modem *RE -S1* as repeater:

1. Select the output **Power** value for the radio modem.
2. Select the **Channel** number to set the one from ten channel to transmit date. In the corresponding rover receiver you need to select the given channel number.
3. Set the operation **Protocol.** Select *FH915 Ext* protocol when the receiver will be used as repeater.
4. Select the **Location** to adjust the frequency range depending on the country.

Radio Parameters for Satel

In this dialog you can configure *Satel* radio modem:

1. Select the output **Power** value for the radio modem.
2. Set the operation **Protocol.**
3. Select an appropriate operation **Protocol** for data transmitting/receiving:
   - *Satel FCS Off* protocol is Satel protocol without Free Channel Scan (FCS).
   - *Satel FCS On* protocol is Satel protocol with Free Channel Scan (FCS).
   - *PDL* protocol can selected for any modem types.
   - *TrimTalk (P)* protocol is TrimTalk protocol, which you need to select for working with any rovers receivers (not Trimble) with the TrimTalk protocol.

- *TrimTalk (T)* protocol is TrimTalk protocol, which you need to select for working with the Trimble rovers receivers with the TrimTalk protocol.

4. Select the type of **Modulation** for the *PDL* protocol only. You can select between *GMSK* and *4-level FSK*.

Radio Parameters for internal and external GSM modems

In this dialog you can configure internal base and rover cellular modem for *Digital UHF* , *Digital UHF II*, *Satel GSM*, *FH 915+ GSM*, *General Internal GSM*,and *Auto* detect device model.

For Base radio:

1. Enter the **PIN** of the Base modem SIM card.
2. Optionally, you can select the bearer service type for base cellular modem. To do this, select the *Show Bearer Service* option from the pop-up menu that displays by clicking  in the top left corner.
3. From the list in the **Bearer Service** field you can select the bearer service type for base cellular modem in the CSD mode. By default, *Don't set up* is selected.

For Rover radio:

1. Enter the **PIN** of the Rover modem SIM card.
2. In the **Base Phone Number** field, enter the base receiver phone number that will be used for the correction transmission.
3. The desired base phone number you can select from the **Phone number list** field. To add the base phone number to the **Phone number list** press the *Add* button. To remove any phone number from the field, select the desired phone number and press the *Delete* button.
4. Optionally, you can select the bearer service type for rover cellular modem. To do this, select the *Show Bearer Service* option from the pop-up menu that displays by clicking  in the top left corner.
5. From the list in the **Bearer Service** field you can select the bearer service type for rover cellular modem in the CSD mode. By default, *Don't set up* is selected.

Modem Dialup

In this dialog you can configure internal/external cellular modem of the receiver for the following GPS+ survey types:

- Network RTK
- RTK: for Base receiver

- MAGNET Relay: for Base receiver
- Network DGPS

Internet Address

In this dialog you can configure settings for Internet connection for the following GPS+ survey types:

- Network RTK
- RTK: for Base receiver
- Network DGPS

Radio Parameters for AirLink CDMA (MUDP)

In this dialog you can configure external cellular modem *AirLink CDMA(MUDP)*:

1. Enter an Internet **Address to Add** that will be used for the connection between the base and several rovers using UDP protocol:.
2. The desired server address you can select from the **Ip Address list** field. To add the entered address to the **IP Address list**, press the *Add* button. To remove any address from the **IP Address list**, select a desired address and press the *Delete* button.

Rover GPRS Parameters

In this dialog you can set the IP address of the base modem for *AirLink GPRS* rover device model.

1. Enter the GPRS address of the base in the **Base GPRS Address** field.
2. The desired base GPRS address you can select from the **IP Addresses list** field. To add the GPRS address to the **IP Addresses list** press the *Add* button. To remove any base GPRS address from the field, select the desired phone number and press the *Delete* button.

Rover CDMA Parameters

In this dialog you can set the IP address of the base modem for *AirLink CDMA* rover device model.

1. Enter the CDMA address of the base radio in the **Base CDMA Address** field.
2. The desired base CDMA address you can select from the **IP Addresses list** field. To add the CDMA address to the **IP Addresses list** press the *Add* button. To remove any base CDMA address from the field, select the desired phone number and press the *Delete* button.

Rover CDPD Parameters

In this dialog you can set the IP address of the base modem for *Generic CDMA* rover device model.

1. Enter the CDPD address of the base radio in the **Base CDMA Address** field.

2. The desired base CDPD address you can select from the **IP Addresses list** field. To add the CDPD address to the **IP Addresses list** press the *Add* button. To remove any base CDPD address from the field, select the desired phone number and press the *Delete* button.

Hybrid Positioning

The Hybrid positioning mode allows you to simultaneously record the point coordinates from GPS rover receiver and Total Station to the opened job. This option can be activated only for Robotic Total Station with reflector. The GPS receiver and Robotic Total Station are connected to the job. The user can separately create own configuration for a GPS receiver and for Robotic Total Station and depending on survey task select either the GPS or Optical instrument by clicking  or  at the home screen.

To activate Hybrid Positioning select the appropriate check box in the Settings dialog. When the Hybrid Positioning is enabled, the icon  is displayed in the home screen.

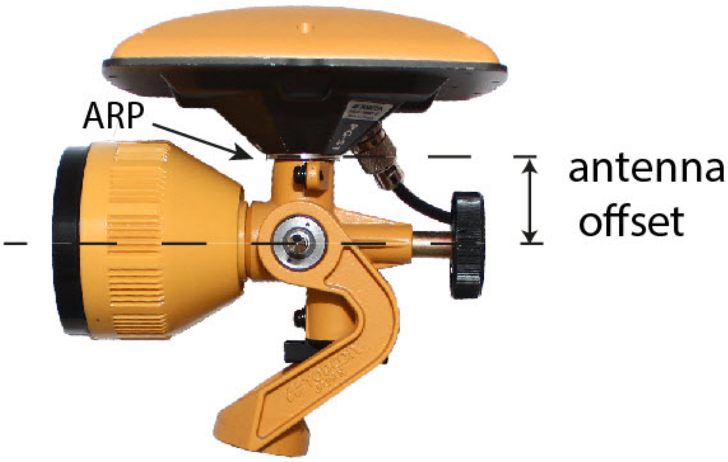
When you activate the Hybrid Positioning, you can:

1. Perform resection with new features. More...
2. Automatically search for an Reflector
3. Automatically perform the localization (for five measured points) and for known parameters between WGS-84 and Local coordinate systems:
   - Load the Bing Maps for the Total Station measurements to the opened job .
   - Send the coordinates measured by Total Station in WGS-84 coordinate system to Enterprise server.

In surveying process, if point coordinates determination are performed by the GPS+ receiver and the optical device at the same time, it is required to mount the GPS antenna and the reflector. One of the possible solutions is the usage of the A7 prism with the special adapter for the GPS antenna or GPS receiver. See the picture below:

For this case you need to select the corresponding prism for Foresight point in the Config:Survey Settings dialog. For the prism the software will automatically use the predefined offset between ARP and horizontal optical axis for calculating of the point elevation when Hybrid Positioning is selected.

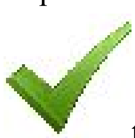
If you do not have "Topcon A7 360" prism, you can create own prism (by selection <Add New> in the field Prism in the Config:Survey Settings dialog), enter offset between ARP and horizontal optical axis and select the **Hybrid Positioning antenna offset** check box.



If you select the **Automatic Localization** check box in the Settings dialog, first **five** points will be used for calculating parameters between WGS-84 and Local coordinate systems. These parameters are saved in the job and they can be automatically updated during next measurements in the Hybrid Positioning (if residuals for these points will be less than for the previous set of points).

Also in the Hybrid positioning mode you can manually select localization points and use Grid to Ground transformation.

If the localization are performed or Grid to Ground transformation is selected or a Grid coordinate system was selected for Total Station measuring:

- the Hybrid Lock icon () is available in SideShot-Direct and Resection 3D dialogs. Clicking on the icon automatically turns the Robotic Total Station to the reflector. To search the reflector, the software has to have the coordinates of the occupation point (where the Robotic Total Station is set) and current coordinate of a point where the rod (with GPS antenna and reflector) is set in WGS -84 coordinate system.
- the Bing Maps check box is available in the General tab of the Map Properties dialog. Select the check box and click  to load Bing Maps to the job.

Resection in Hybrid Positioning

Before performing resection you need activate **Hybrid Positioning**, select desired prism and select **Hybrid positioning antenna offset** check box and open **Resection** dialog.

There are four different scenarios to perform the resection procedure:

1. The desired Grid projection is defined in the job. Grid to Ground transformation is not selected. The GPS and TS measurements are performed for each unknown point. The coordinates of the Occupation Point are calculated in the current Grid coordinate system.

   You measure first control point by Robotic Total Station, save it in the job and the software prompts to measure GPS coordinate of the point and save it. After measuring two sets of coordinates for first point the software prompts specify next control point. After saving second point the result screen displays total station measured value of angles and slope distances with estimate of accuracy. You can continue to measure a other point (clicking **Add** button) or finish the measuring and calculate the occupation point coordinates using two points (clicking **Accept** button) in the current Grid coordinate system. More...

2. The desired Grid projection is defined in the job. Grid to Ground transformation is selected. The relation ship between Grid and Ground coordinate systems is known. Coordinates for both coordinate sets are displayed in the current coordinate system. The coordinates of the Occupation Point are calculated in the Grid or Ground coordinate systems.

   The routine of the resection procedure is equal scenario 1. More...

3. A Grid projection is not defined in the job. The user has coordinates of the control points (two points at least) in the Ground coordinate system. For each control point TS and GPS measurements are provided. The software automatically performs the localization between WGS-84 and Ground. The coordinates of the Occupation Point are calculated in the Ground and WGS-84 coordinate systems.

   You need either to enter a new point name and manually enter the desired coordinates or select from the list, save the TS measurements for the point in the job. The software prompts to measure GPS coordinate of the point and save it. Then the software prompts specify another control point. After saving second point the result screen displays total station measured value of angles and slope distances with estimate of accuracy. You can continue to measure a next point (clicking **Add** button) or finish the measuring and calculate the occupation point coordinates using two points (clicking **Accept** button). After calculating the occupation point coordinates in the Ground coordinate system, the software automatically performs the localization between WGS-84 and Ground using the measured points. More...

4. The coordinates of the control points (two points at least) in the Ground coordinate system imported in the job. The desired Grid projection is defined in the job. For each control point TS and GPS measurements are

provided. The software automatically performs the localization between Grid and Ground. The coordinates of the Occupation Point are calculated in the Grid and Ground coordinate systems.

You need to import the set of points with coordinates in Ground coordinate system to the job. Then select the desired Grid projection. After that, the select from the list desired point, save the TS measurements for the point in the job. The software prompts to measure GPS coordinate of the point and save it. Then the software prompts specify another control point. After saving second point the result screen displays total station measured value of angles and slope distances with estimate of accuracy. You can continue to measure a next point (clicking **Add** button) or finish the measuring and calculate the occupation point coordinates using two points (clicking **Accept** button). After calculating the occupation point coordinates in the Ground coordinate system, the software automatically performs the localization between Grid and Ground using the measured points.

More...

Optical Survey

To create a new configuration:

1. Enter the **Name** for the configuration that will be displayed in the Configurations list.
2. Only one survey **Type** is available for configuration:

   Robotic - survey can be performed by one person with a motorized instrument.

3. Click **Next**. The wizard will set default parameters specific for the equipment selected.

Instrument

Set an instrument for the survey and click **Next**:

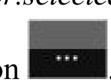
1. Select the **Manufacturer** of total stations. MAGNET Field Site supports Topcon and Sokkia.
2. Select the total station **Model**. By default, the PS Series are selected.
3. You can click **Peripherals** to configure available peripherals in the dialogs later. More...
4. Optionally, you can set parameters for Monitor survey. To do this, click the button  and select Monitor from the pop-up menu that will appear.

Robot Peripherals

If required, select only available **Depth Sounder** check box to configure a depth sounder. Click **Parameters** to specify settings. More...

Monitor Options

To configure settings for data output in the Monitor survey:

- From **Log To**, select None, File, or COM1 for data output.
- Select the **Type** of data to output: *Raw Data* or *Coordinates*.
- Select the **Format** from FC-5, FC-6/GTS-7 or GTS-6 for raw data output, or GGA to output coordinates.
- If *File* is chosen to log to, the **File Name** field shows the default file name (*Monitor.selected format*) and location of the file to log into. To change the file name and its location, click the button .
- If a serial **COM1** port is selected, enter the baud rate, parity, data and stop bits settings for the communication port.

- Select the **Store As Check Point** check box to store the measurements as Check Points to the original points in the monitor list; otherwise, they will be stored as normal points. This option is the only available when you select *None* from the Log To drop-down list.

Connection mode

Select the type of connection with the instrument. You can connect your field controller with the instrument through the *Cable*, *Radios*, *Bluetooth* or *RC-devices.*

For PS Series, click the **RC-Handle attached** check box if such an equipment is attached.

Cable

Configure parameters of the cable connection: **Baud** (baud rate), **Parity**, **Data** (number of the data bits), and **Stop** (number of the stop bits).

Connection with Robot

Configure a connection of the field controller with the equipment in the Robotic survey and click **Next**:

1. Select the type of connection with the **Total Station** from the following: *Cable, Radios, Bluetooth, RC3 Only,* or *RC4 Only (SS Wireless).*
2. Select the type of connection with the **Remote Controller** device from *RC3 Cable, RC3 Bluetooth, RC4 Cable, RC4 Bluetooth,* or *None* as required, depending on the type of connection with the total station.
3. Select an appropriate channel of communication of the RC device with the total station.

Radio

Set communication parameters of the radio:

1. Select the **Type** of the radio: Satel, Pacific Crest, or Generic.
2. Configure **Radio Communication Settings**: Baud (baud rate), Parity, Data (number of the data bits), and Stop (number of the stop bits).
3. Click the **Configure Radio** button to configure the parameters of the radio. The dialog depends upon the radio chosen.

Search/Track (Topcon)

Configure the settings for the Topcon total station signal tracking in the Robotic mode:

1. Set the **Turning Speed** of a total station, in revolutions per minute.
2. In **Start Search After**, set the delay between the loss of the signal and the start of searching.
3. Set the **Pattern** of program for the tracking and searching.
   - The *Normal* (for AP-L1A and GRT-2000) or *Pattern 1* (for the rest of the instruments) mode can be selected to search the prism at the point where the prism is lost. The Instrument searches in up and down directions gradually from the point where the prism is lost. The searching continues until the prism is found.
   - The *High* (for AP-L1A and GRT-2000) or *Pattern 2* (for the rest of the instruments) mode also can be selected to search for the prism. In this case the search pattern tries to locate the prism in a very short time. Instrument searches from up to down. The searching continues until the prism is found or a maximum of six attempts.

     Note: The auto tracking mode changes to manual mode when the prism cannot be found within six attempts, and returns to the point where the prism was lost.
4. Select the mode for the **Tracking Speed** as required according to the purpose of measurement: *Survey* (for fixed point observation), *Machine Control 1, 2* (to control construction machinery), or *Slow*, *Medium*, and *Fast* (for real-time surveying of various traveling objects with AP-L1A and GRT-2000 total stations).
5. From **Sensitivity***,* select the power level of the accepted signal. It can be low, medium or high. Available for selection only in AP-L1A and GRT-2000 total stations.
6. From **Track Light**, select On or Off to switch on/off the light on the line of sight.
7. Set the **Scan Range** of the tracking signal. It can be Narrow, Middle or Wide. Available for selection only in AP-L1A and GRT-2000 total stations.
8. Set the **Search Area** of searching, in degrees, for the vertical and horizontal planes.

Search/Track (Sokkia)

For Sokkia robotic total stations, only set the **Search Area** of searching or tracking, in degrees, for the vertical and horizontal planes.

Survey Settings

Configure the settings that will be used by default during the survey and click **Next**.

1. Select the **Measurement Method** of side-shot measurements from: Sideshot-Direct and Sideshot Direct/Reverse.

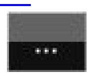
2. In **Tolerances**, set the admissible deviation values of the horizontal and vertical angles and the distance in the current units.
3. The **Auto Accept Meas** option is selected, when the total station turns to the prism, it will automatically take a measurement, then turn to the next point in the sequence. If it is not selected, once the total station turns to the prism, you will be prompted to accept the measurement, then turn to the next point in the sequence.
4. If required, select the **Measure Reverse Dist** check box to enable the reverse distance measurements in conventional and reflectorless surveys. These are used for the reduction of the distance measurement errors.
5. Select the **Num Meas to Avg** check box to set a number of measurements to be averaged in the near field.
6. Optionally, you can define the point increment value for Survey points. To do this, click  and select the Point Properties option from the pop-up menu that appears.

Point Properties

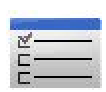
Set a value to increment/ decrement the survey point number when adding a new point.

Survey Settings

Continue configuring the survey. The settings will depend on the selected configuration type and instrument:

1. From **Meas Type**, select the order and the type of the measurements in one set. Here: *HA* - horizontal angle; *VA* - vertical angle; *SD* - slope distance; *HD* - horizontal distance; *VD* - vertical distance.
2. From **Backsight P.C.** / **Foresight P.C**., select prisms for backsight and foresight measurements.
   Note: The backsight prism is considered when the backsight distance is measured and the backsight reflector height is fixed. More...
3. If required, click the  button to Edit Prisms.
4. From the **Target Type** drop-down list, select one of the available methods of distance measurement with EDM, for instance, Prism, Sheet, Non-Prism or Long Non-Prism. The selection depends on the instrument you use.
5. In the **Auto Topo** field, select the following parameters to automatically store positions in the Robotic survey:

- From the *Method* drop-down list, select the method for setting the interval between the measurements: by time, by horizontal distance, or by slope distance.
- In the *Interval* edit field enter the value of that interval.

Note: Every survey parameter can be changed with the help of the  button from any Survey screen in Optical mode.

Edit Prisms

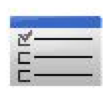
This dialog enables you to edit a prism pre-defined in MAGNET Field Site and define a new prism:

1. Select the **Prism** name from the drop-down list.
2. In the **Prism Info** field that displays the pre-defined values for the selected prism, you can edit the *Name* and the value of the *Constant*. Prism Constant is the parameter of the prism characterizing the difference between the reflection plane and the center of the prism. Select the *Sokkia 360 Prism* check box if you are using such a prism.
3. Click the **Update** button to save the changes you made. To save a new prism, click the *Add* button.
4. Click the **Delete** button to delete the selected prism from the list.

Survey Settings

To configure settings that will be used during a Robotic survey:

1. In the **Precise** area, configure storing positions:
   - From the **EDM Mode** drop-down list, select the sensitivity to use for the distance measurements. By default, it is set to *Fine 1mm*.
   - Select the **Meas Continuously** check box to log measurements continuously and stop them manually when required.
   - The **Average** field allows you to set a specific number of measurements for logging and average them for storing the position. If required, you can change the default number (3) of measurements.
     Meas Continuously is enabled, the logging immediately stops to store the point.
     Meas Continuously is disabled, the **Average** position is calculated to store the point.
   - Select the **Auto Store** check box to enable auto-storing positions for the average mode.
2. In the **Quick** area, configure fast auto-storing positions:
   - From the **EDM Mode** drop-down list, select the sensitivity to use for the distance measurements. By default, it is set to *Coarse 10mm*.
   - In the **Average** field, if needed, you can change the default number (1) of measurements required for storing positions.

Note: Every survey parameter can be changed with the help of the  button from any Survey screen in Optical mode.

Stake Settings

Configure settings that will be used during a stakeout.

1. In **Horizontal Distance Tolerance**, enter the horizontal distance tolerance value when the graph switches to a bull's eye in Stakeout.
2. Select the **Screen Orientation** to be used during the stakeout. Moving Direction +North is similar to the Moving Direction option, but displays the North direction when you are within three meters of the design point.
3. From **Display Reference**, select a reference orientation for the display.
4. From **Turn TS to Design Point**, select the method which the total station will use to turn toward the design point.
   - If you select *Display HA only* (or *Display HA/VA*), the total station will display the computed values of angles to design points, and you will turn the instrument by these angles. A motorized instrument can track the prism and turn after it.
   - If you select any *Automatic* method for a motorized instrument, the instrument will automatically turn to the design point after storing the first staked point.
5. Select the **Search After Turn** check box to cause the instrument to search for the prism after the instrument automatically turn to the next design point.
6. Select the *Display* option from the pop-up menu that appears by clicking  in the top left corner of the dialog if required to specify an icon for the staked point. More...

Miscellaneous (Optical)

The Miscellaneous dialog allows you to customize the user interface:

- **Display coordinates after measurement**: when selected, computed coordinates are displayed automatically after a TS measurement is performed and before the point coordinates are stored into the database.
- **Apply Earth Curvature and Refraction**: when selected, the computed heights for Earth Curvature (Vertical Distance) and slope distances and vertical angles are corrected for atmospheric refraction.
- **Prompt for Rod Height**: when selected, a prompt for a height of the Rod (Target) appears before a point is stored.
- **Right/Left Offset: Rod to TS**: when selected, the right and left offsets are shown in the direction from Rod to TS.
- **Stakeout Sound**: when selected, a sound in Stakeout indicates as the point is staked.
- **Beep on Storing Points**: by default this is selected to beep each time the point is stored.
- **VA Zero at Level/Horizontal**: this is not selected by default and VA is set to zero at Zenith. When the check box is selected, VA is set to zero at the Level/Horizontal direction. Only certain total stations allow

MAGNET Field Site to set this value. For this reason, ensure that this option is set to the same value in the total station.

- **Use Horizontal Angle Left**: when selected, the horizontal angle measurements are shown in a anticlockwise (Left) direction. If this check box is not selected, the horizontal angle measurements are shown in a clockwise (Right) direction.
- **Automatically display BS Setup screen**: when selected, the Backsight Setup Dialog will be displayed automatically when you attempt to access any of the dialogs involving total station observations.
- **Hold Offset Measurement**: when selected, the dialog to measure an offset point with the help of the selected offset tool displays automatically after each measurement.
- **Remember Occ/BS if set**: when selected, you will not have to set the BS again if it has been set once for total station and level observations.

Coordinate System

To specify the Coordinate System for the Job:

1. In **Projection** select the projection to be used from the drop-down list. Initially the list is empty. Click to activate projections necessary for work. More...
2. Select the **Use NADCON** check box, available for all projections on NAD27 datum, to make the coordinates in MAGNET Field Site equal to the same from Corpscon (Coordinate Conversion Software).
3. Select the **Use Grid/Ground** check box to enable transformation to ground coordinates for surveying. Click to set transformation parameters. Find out more on Grid to Ground transformation.
4. In **Datum** specify the datum as required for the projection selected. Select the datum from the drop-down list.

   Note: The NAD83 datum has three independent realizations in MAGNET Field Site with respect to the WGS84 datum: NAD83, NAD83(ITRF96) and NAD83_NO_TRANS. More...

   Click to add a custom datum if necessary. More...

5. Select **Geoid** from the drop-down list. The job will refer to the selected geoid file to convert ellipsoidal heights to elevations. Click to add the geoid to the list. More...

Projections

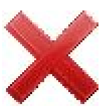
The Projections dialog contains a list of cataloged projections divided by regions, that can be chosen for use in the job.

To add a desired projection to the list in the Coordinate System dialog:

In the **Pre-Defined** panel:

- Click the region node to expand the tree of available projections, and use the scroll bar to view the full list of projections.
- Highlight the name of the desired projection.
- Use the arrow button to select the chosen projection in the Pre-Defined panel and insert it into the Active panel.
- Repeat the steps to add other projections.
- Create a custom projection to the list of pre-defined ones by clicking the **Custom** button. More...

In the **Active** panel:

- View all active projections.
- Use  to delete a highlighted projection from the Active panel if required.
- Click  to add the active projections to the drop-down list in the Coordinate System dialog.

Grid/Ground Parameters

To set Grid/Ground parameters:

1. In **Parameters**, select which set of parameters to enter: *Scale factor (more...), Average Job Height (more...) or Origin Point (more...)*.
2. In Origin Pt and Scale Factor modes directly enter the **Combined Scale Factor** to be used in the transformation. In any Parameters mode resulting combined scale factor will be displayed at the bottom of the dialog for reference purposes.
3. In Avg Job Ht mode enter the **Map Scale Factor** from Ellipsoid to Grid to be used in the transformation.
4. In Scale Factor mode designate **combined scale factor transformation direction**. In Origin Pt and Avg Job Ht modes this *displays used direction*.
5. Enter the **Average Job Height** to be used in the transformation.
6. Enter the angle value of Azimuth Rotation directly if it is known.
7. Click to calculate the rotation from azimuths on Grid and Ground. More... Azimuths/Bearings in turn can be calculated from points in the job. More...
8. In Scale Factor or Average Job Ht modes enter **Offsets of the origin along the North and East axes**. These offsets are from the Geodetic North -> Ground North.
9. In Origin Pt mode select from map, list or enter **the coordinates for the Origin Point**.
10. In Origin Pt mode enter **the coordinates for the Ground Point**.

Learn more... about how to perform Grid/Ground transformation.

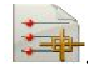
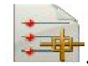
Compute Rotation

To calculate Rotation from azimuths on Ground and Grid:

- You can either manually enter the **Grid** and **Ground Azimuths** or click the Compute button to calculate azimuth using points in the job that define the needed directions.
- Click to compute Rotation.

Compute Azimuth/Bearing

To compute the azimuth by two points that define the direction:

1. In **From**, enter the name of the initial point or select the point from the map  or the list .
2. In **To**, enter the name of the terminal point or select the point from the map  or the list .
3. Enter an angle to **Add to Azimuth** if required.
4. View the computed Azimuth.

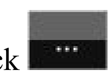
Custom Projections

This dialog contains a list of custom (user-defined) projections.

- Initially the **Projection** list is empty.
- Click the **Add** button to create a new custom projection and add it to the list. More...
- Click the **Edit** button to change a projection that was added to the list.
- Click the **Delete** button to delete a highlighted projection from the list.

Custom Projection

To create a new custom projection:

1. Enter a **Name** for the new projection.
2. Select a **Type** from the list of sample projections.
3. Select a **Datum** from the displayed list of datums, or click  to create a custom one (more...) and then select it.
4. Enter the **Region** to which the projection belongs. If no region is specified, it will be added to the Global list.
5. Enter a description (**Note**) for the projection.
6. Click **Next** to enter the new projection specifications depending on the selected sample projection type. More...

Custom Projection

Depending on the Type of projection selected, you are prompted to fill in some of the following fields.

1. Enter the latitude (*Lat0*) and longitude (*Lon0*) of a point chosen as the origin on the central meridian of the custom projection.
2. Enter the false Easting (*East0* or *E0*) and Northing (*North0* or *N0*) of the origin for the projection. These constant values are added to all negative Eastings and Northings to get only positive values of Easting and Northing.
3. Enter the *Scale* that sets a constant scale factor along the central meridian of the custom projection.
4. Enter the longitude of *Central Meridian* for the projection.

5. Enter the North and South latitudes bounding the area of the custom projection (*NorthLat* and *SouthLat*).
6. Enter the azimuth of the axis (*AxisAzimuth*) for the projection.
7. Click  to create the new projection and add it to the Projection list.

Note: The Latitudes are positive for the Northern Hemisphere, and negative for the Southern Hemisphere. The Longitudes are positive for Eastern directions and negative for Western directions relative to the GMT line.

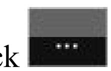
Custom Datums

This dialog contains a list of custom (user-defined) datums.

- Initially the **Datum** list is empty.
- Click the **Add** button to create a new custom datum and add it to the list. More...
- Click the **Edit** button to change a datum that was added to the list.
- Click the **Delete** button to delete a highlighted datum from the list.

Custom Datum

To create a new custom datum:

1. Enter a **Name** for the new datum.
2. Select an **Ellipsoid** from the displayed list of ellipsoids, or click  to create a custom one (more...) and then select it.
3. Enter a description (**Note**) for the datum.
4. Click **Next** to enter transformation parameters to convert from WGS84 to the new datum. More...

Custom Datum

MAGNET Field Site uses the seven-parameters Helmert Transformation Strict Formula for datum transformation.

To create a custom datum:

1. In **Offsets**, enter the Datum to Space (WGS84) offsets: dX, dY, dZ.
2. In **Rotations**, enter the Datum to Space (WGS84) rotations: rX, rY, rZ, in seconds.
3. Enter the **Scale** in ppm.
4. Click  to create the new datum and add it to the Datum list.

Custom Ellipsoids

This dialog contains a list of custom (user-defined) ellipsoids.

- Initially the **Ellipsoid** list is empty.
- Click the **Add** button to create a new custom ellipsoid and add it to the list. More...
- Click the **Edit** button to change an ellipsoid that was added to the list.
- Click the **Delete** button to delete a highlighted ellipsoid from the list.

Custom Ellipsoid

To create a custom ellipsoid:

1. Enter a **Name** for the ellipsoid.
2. Change the values of ellipsoid semi-major equatorial axis (**A**) and inverse flattening (**1/F**) as required.

   Note: By default, the *A* and *1/F* fields contains the values for the most common reference ellipsoid defined by WGS84.

3. In **Note**, enter a description for the ellipsoid.
4. Click  to create the new ellipsoid and add it to the list of all the available Ellipsoids.

Geoids List

Geoid is the physical reference surface of the Earth. Its shape reflects the distribution of mass inside the Earth. Geoid undulations are important for converting GPS-derived ellipsoidal height differences to orthometric height differences (elevations).
The Geoids List dialog contains a list of Geoids available for selection: the *Name* and *Full Path*

- Initially the **Geoid List** is empty.
- Click the **Add** button to add a new geoid to the list. More...

  Note: Install the geoid file on the disk prior to adding it to the list. Some geoid files can be installed into the Geoids folder during MAGNET Field Site installation. They are provided with the installation program as Geoid File Format (.gff) files.

- Click the **Edit** button to change a geoid that was added to the list.
- Click the **Remove** button to delete a highlighted geoid from the list.

Add/Edit Geoid

In the Add/Edit Geoid dialog, select a Geoid file from disk and see the boundaries for the geoid. After being chosen, the geoid file appears in the Geoids List dialog.

To add a geoid to the geoids list:

1. Select the **Geoid Format** of the geoid file to load.
2. Click Browse to navigate to the desired file on the disk, and select it.
    - After the geoid is chosen,  displays the path to the file.
    - After the geoid is chosen, you can see the boundary of the geoid application: the longitudes and latitudes of the north-west and south-east corners of the geoid boundary.
3. Click  to add the selected geoid to the list of all the available Geoids in the jobs.

NAD83 Datum Details

The NAD83 datum has three independent realizations in MAGNET Field Site with respect to the WGS84 datum: NAD83, NAD83(ITRF96) and NAD83_NO_TRANS to reflect the updates to these datums. The original intent was for WGS 84 and NAD 83 to be identical. The mathematical definition of the ellipsoids (WGS 84 and GRS 80) differs slightly due to the choice of defining constants and number of significant figures. The maximum discrepancy between a Cartesian X,Y,Z coordinate projected onto both ellipsoids is 0.1 mm at 45 degrees latitude. So consider the ellipsoids to be identical.

There is some confusion between the WGS 84 and NAD 83 datums. When NAD 83 was first realized in 1986, it used the same control stations as WGS 84, some of which were Doppler stations, which were accurate to about one meter. Consider the datums to be identical.

Since this time, there have been several realizations of WGS 84, the latest being WGS 84 (G1150), which was performed using data from IGS tracking stations collected during GPS Week 1150. Recent studies have shown that WGS 84 (G1150) is essentially identical to the International Terrestrial Reference Frame of 2000 (ITRF00). Also, during this time there have been no new realizations of NAD 83.

What this means is that WGS 84 and NAD 83 can no longer be considered identical and are in fact different by more than one meter. This is because the WGS 84 datum has been updated over time, using GPS while the NAD 83 datum has remained constant since 1986. However, most software manufacturers still consider WGS 84 to be identical with NAD 83.

To be compatible with other manufacturers, Topcon provides transformation parameters from WGS 84 to NAD 83 where all the parameters are zero. This means WGS 84 equals NAD 83. This set of datum transformation parameters is called NAD 83 No_Trans.

Topcon also provides another set of datum transformation parameters called NAD 83 which reflects the updates to WGS 84. These parameters are taken from the National Geodetic Survey.

Topcon only uses the first seven parameters, three translations, three rotations, and scale.

Global Settings

Global Setting are used by all of the jobs. Changes made in Global Settings work on all of the jobs.

In the Global tab:

1. Select the **Use Bold Font** check box to display the text in bold typeface.
2. Select the **Enable Job History** check box to enter and save the surveyor's operation on the job in the file.
3. If required, select the **Port Data Logging** check box to record traffic with the currently connected device into a txt file. After you click , you have to reconnect with the device. More...
4. From **Color Scheme**, select a color for screen background if you want to change the default dark.

In the New Jobs tab, set parameters to automatically import:

1. **Localization from previous job**:
   - Select *Never import* to create new jobs without previous localization.
   - Select *Always import* to automatically export localization from the last open job to a new job upon opening the new job.
   - Select *Prompt to import* to choose if you want to use localization from the last open job.
2. **Global code library to job**:
   - Select *Never import* to create new jobs without automatic import of global codes.
   - Select *Always import* to automatically import.
   - Select *Prompt to import* to choose whether to import or not.
3. To **Continue point name numbering** in the job, select this check box.

In the Keyboard tab:

1. **Button Sound** is the default to provide sound effects when you click any functional button.
2. **QWERTY Keyboard** is the default to enable the QWERTY layout of the soft keyboard. If you clear this check box, the ABC keyboard is used.
   Note: For PC version always QWERTY keyboard is used.

Job Backup

Backup copies are automatically created for the current job and safely stored with new names "<job file_ name>!YYY-MM-DD!. mjf.bak" in the *Target folder*. By default, backup files are stored in the Jobs folder. You are able to open the backup of the job file in the usual way. More...

To configure backup settings:

1. You can change the target folder. To do this, select the **Custom** check box, click the *Browse* button and select the backup target folder.
2. Select the **Frequency** in which you want the backup to occur. Ten minutes is the default. If you select None, backups will not be created.
3. In **History Depth**, change the number of backups to keep if required. Three files is the default.
   Note: MAGNET Field Site will create a separate *.bak file for the current job every time you open the job file during the day with another date. If the job file is opened in subsequent days, the *.bak files previously formed will be overwritten to the ones with the newer dates.
4. Click to save the settings and return to the Home screen.

Instrument Settings

Depends on the selected Total Station model the dialog has one or two tabs. For IS310 model the dialog contains two following tab:

Temperature / Pressure tab and Config tab. For other models the dialog contains only Temperature / Pressure tab.

Temperature / Pressure tab

To configure the air conditions around the instrument for calculating the atmospheric correction of measured distances:

1. Enter the value of the **Temperature** in the current units.
2. Enter the value of the air **Pressure** in the current units.
3. **PPM** shows the relative distance error calculated for these values.
4. Click the **Send to Instrument** button that appears only for supported instruments (for instance, PS Series that have commands to set temperature and pressure) to save the data in the instrument. For the instruments not supported, a message will prompt you to manually enter these values on the instrument.

Config tab

In this tab you can:

- select power source for total station: internal (Li-ion) or external (12V battery).
- activate the tilt sensors in the horizontal or horizontal and vertical planes in the total station.

To configure the air conditions around the instrument for calculating the atmospheric correction of measured distances:

1. Enter the value of the **Temperature** in the current units.
2. Enter the value of the air **Pressure** in the current units.
3. **PPM** shows the relative distance error calculated for these values.
4. Click the **Send to Instrument** button that appears only for supported instruments (for instance, PS Series that have commands to set temperature and pressure) to save the data in the instrument. For the instruments not supported, a message will prompt you to manually enter these values on the instrument.

Code Options

This dialog allows you to configure:

Quick Codes

Settings for Codes

Prompts for Codes

Quick Codes

Quick Code is a code that appears in a box on the Map and allows you to take measurements with this code in Quick mode in Topo survey and to log now in Auto Topo survey by clicking the box. Up to six codes are available for such configuration at a time.

To configure quick codes:

1. Select a check box.
2. Enter the name of the desired code. You can type in the name of an existing code or select it from the drop-down list. If you type in a new name, the Code dialog will prompt you to create a new code.
3. For a line or area code, enter a string value.

Code Settings

The Settings tab allows you to configure global settings for codes:

In the Codes field, configure the settings for codes:

1. In **Default New Type**, set the default type for a new code. If set to *Prompt*, you will be prompted to define the new code when storing points.
2. In **Data Entry**, set the preferred entry mode between *Notes* and *Codes* for Survey dialogs.
3. In **Code File**, click the Browse button to select the Global Code file to be used along with the codes in the job.
   Note: The default code file (MAGNETDefCodeLib.xml) is installed automatically in the *tpsdata* folder upon MAGNET Field Site installation.
4. The global setting **Code with Description** toggles the display of descriptions with Codes.

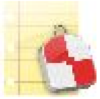
In the Control Codes field, configure the settings for control codes:

1. Selecting **Alow Custom** allows you to custom define the control codes and enables you to set them persistent for Survey dialogs.
2. To set custom control codes persistent for Survey dialogs, select **Allow Persistent** check box.

Code Prompts

In the Prompts tab, select the appropriate boxes in the **Prompt for Code in** field to be prompted for setting codes when storing points in:

Optical Survey
GPS Survey
Stakeout
COGO

 Stake Reports

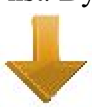
Contains a list of default configurations of stakeout reports and their types.

To edit the list of report configurations:

1. Highlight a name of the report configuration to control.
2. Click the **Delete** button to remove the report configuration from the list.
3. Click the **Edit** button to change the highlighted report configuration. More...
4. Click the **Add** button to create a new configuration. More...

Report Configuration

You can edit:

1. The **Name** of the report configuration. To do this, click in the field.
2. The **Type** of the report configuration. Select a type from the drop down list.
3. Select the corresponding items in the list to include corresponding information in the report. Use the scroll bar to view the entire list. By default all items are included.
4. Use  and  arrow buttons to change the order of displayed information in the report.
5. Click the **Edit** button or just click the item to open a field to edit the item *Name* as required. You can use *Calculator*  to make some calculations if required.

Cloud Connections

To configure settings for communication with the MAGNET Enterprise web-server:

In the *Login* tab, enter the appropriate login information:

- In **Login**, enter the user name of your account.
- In **Password**, enter the secret word of your account.
- Select the **Connect on startup** check box to connect upon starting the program.

In the *Upload* tab:

- Select the corresponding check boxes to specify which reference data related to the current job will be exported.
- If required, clear the **Upload current job by default** check box not to export the job.

In the *Download* tab:

- If required, clear the **Prompt to import file to Job** check box to import files without prompts.

In *SiteLINK 3D* tab:

- If required, select the **Enable SiteLINK 3D features** check box to enter credentials for SiteLINK 3D communication:
- Select the **Connect on startup** check box to connect upon starting the program.

Exchange folder

You can exchange data between the current job and another job, different files of the predefined formats or your own custom formats, and the MAGNET Enterprise project.

Click an icon to perform the task:

To 3DMC
Exports data from the current job to a 3DMC project.

From 3DMC
Imports data from a 3DMC project.

To Job
Exports data from the current job to another job.

From Job
Imports data to the current job from another job.

To File
Exports data from the current job to a file.

From File
Imports data from a file to the current job.

Enterprise Upload
Uploads job data to MAGNET Enterprise.

Enterprise Download
Downloads data from Magnet Enterprise.

 Export To Job

To export data from the current job to another job:

1. Select an existing job or create a new one to which you will export data. More...
2. Define the general types of data to be exported. More...
3. Select what specific data of the defined types will be exported. More...
4. Filter points for export if required. More...
5. View the export progress. More...

Select Job

To select a Job to open:

- Highlight the job in the job list. The job list contains the **Job Names** of all existing jobs created/opened using this software.
- When a job is selected in this list, the **Created**: and **Modified**: fields will reflect when the job was created and last modified.
-  indicates the path to the selected job. By default, the job files are stored in the [program]\Jobs folder.
- If you do not see the desired job name in the list, click **Browse**. It takes you to the Browse dialog to browse directories to search for the job.
- You can export data to a new job. Click **New** to create a new job.

Export Data To Job

To select data to be exported from the current job to another job:

1. From the **Points** drop-down list, select the filter for points to export:
   - All Points
   - By Point List(s)
   - By Type(s)
   - By Range and Code(s)
   - By Type(s), Range and Code(s)
   - or None

   If more than one filter is selected, they will be processed together and only points that meet the settings of all the filters will be exported. Select *None* you do not want to filter.

2. Select the corresponding check boxes to select data types, which should be exported. The data that absent in the current job will be unavailable for selection. More...
3. Click **Settings** if required to setup export of points as control. By default, they are exported as design.
4. Click **Next** to filter the points if required, then choose the needed objects from a list of objects of the selected data type.

Filter Points By Range and Code(s)

To select for exchange points by a range and code(s):

- Select the **Filter by Codes** check box to make the editable field active. Enter the desired codes either manually or click **Select** and choose the codes from a dialog which appears. Select Codes For Filter
- Select the **Filter by Range** check box, to make the editable field active. Specify the names of the points to be included. These can be specified by a range or by enumeration individually. The symbols ';', '.' or ',' can be used for *Name Separator* and '-' for *Range Separator*.
- If available, click **Next** to continue selecting data.
  Note: The Next button is available until all data of chosen types is selected. After the Next button becomes grayed out, the  button appears to start the export process.

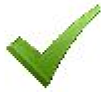
Select 'Object' to Export

The title of this dialog changes based on the data type selected for export.

1. When filtering points by types, the existing types of points will be available for selection.
2. The objects in this dialog are those available in the current job.
3. Choose the needed objects from a list of objects:
   - Selection can be done by placing check marks in the list next to the desired codes.
   - Click **Check** and **Uncheck** to toggle the highlighted item(s) on and off, respectively.
   - Click **Select All** to choose all the items at a time.

   - Selection can be done by the context menu that depends on the place where it pops up:

     - **Select All:** highlights all the types in the list.
     - **Select All Below:** highlights all the types below the highlighted line.
     - **Select Several**: highlights the required types.
     - **Cancel Selection:** removes highlights in the list.
     - **Check:** places check marks in the highlighted lines.
     - **Uncheck:** clears check marks in the highlighted lines.

4. If the **Next** button is available, click it to continue selecting data.
   Note: The Next button is available until all data of chosen types is selected. After the Next button becomes greyed out, the  button appears to start the export process.

Select Codes For Filter

All the codes existing in the current job will be listed in the Code(s) list.

Select the code(s) for which you want to exchange all the points having that code:

- Selection can be done by placing check marks in the list, against the desired codes.
- Click **Check** and **Uncheck** to toggle the highlighted item(s) on and off, respectively.
- Click **Select All** to choose all the items at a time.

Export Status

The export process is reflected in the Export Status dialog that contains a progress bar and comments about the export process. The progress bar displays the percentage of the points being exported.

Export Warnings

Displays the warning messages during the Export process if necessary.

Import From Job

To import data from another job to the current job:

1. Select the job from which you will import data. More...
2. Define the general types of data to be imported. More...
3. Filter points for import if required. More...
4. Select what specific data of the defined types will be imported. More...
5. View the import progress. More...

Import Data From Job

The title of this dialog contains the name of the job selected. The dialog enables you to select the data to import and, if necessary, filter the imported points:

1. From the **Points** drop-down list, select the filter for points to import:
   - All Points
   - By Point List(s)
   - By Type(s)
   - By Range and Code(s)
   - By Type(s), Range and Code(s)
   - or None

   If more than one filter is selected, they will be processed together and only points that meet the settings of all the filters will be exported. Select *None* you do not want to filter.
2. Select the corresponding check boxes to select data types which should be imported. The data that absent in the current job will be unavailable for selection. More...
3. Click **Settings** if required to setup import of points as control. By default, they are imported as design.
4. Click **Next** to filter the points if required, then choose the needed objects from a list of objects of the selected data type.

Select Object to Import

The title of this dialog changes based on the data type selected for import.

1. When filtering points by types, the existing types of points will be available for selection.
2. The objects in this dialog are those available in the current job.

3. Choose needed objects from a list of objects:
    - Selection can be done by placing check marks in the list next to the desired codes.
    - Click **Check** and **Uncheck** to toggle the highlighted item(s) on and off, respectively.
    - Click *Select All* to choose all the items at a time.
4. If the **Next** button is available, click it to continue selecting data.
   Note: The Next button is available until all data of chosen types is selected. After the Next button becomes grayed out, the  button appears to start the import process.

Filter Points By Range and Code(s)

To select for exchange points by a range and code(s):

- Select the **Filter by Codes** check box to make the editable field active. Enter the desired codes either manually or click Check and choose the codes from a dialog which appears. Select Codes For Filter
- Select the **Filter by Range** check box, to make the editable field active. Specify the names of the points to be included. These can be specified by a range or by enumeration individually. The symbols ';', '.' or ',' can be used for *Name Separator* and '-' for *Range Separator*.
- If the **Next** button is available, click it to continue selecting data.
  Note: The Next button is available until all data of chosen types is selected. After the Next button becomes greyed out, the  button appears to start the import process.

Import Status

The import process is reflected in the Import Status dialog that contains a progress bar and comments about the import process. The progress bar displays the percentage of the points being imported.

Import Warnings

Displays the warning messages during the Import process if necessary.

Export To File

To export data from the current job to a file:

1. Select the **Data** type to export and specify the file **Format** to create as required. More...
2. Set a name and destination directory for the file.
3. Specify the coordinate system and coordinate type for exported points.
4. Define settings for custom text formats. More...
5. View the export progress. More...

Export Data To File

To export different data types to files of either predefined or custom formats:

1. Select the **Data** type to export to the file.
2. Select the **Format** of the file to create for the data type you selected. Each format is intended for storing a certain data group.
3. If required, select the **Select file units** check box to configure units for values in the file. More....
4. If more fields are displayed for some data types, complete them if required:

Points	Raw Data
Lines	Roads
Areas	Localization
Point Lists	Surfaces

Note: When exporting Multiple data to file, the fields for all included data types will be displayed to configure.

Points To File

Configure settings to export points from the current job to a file:

1. Select the **Select types of the points** check box if not all types of points should be exported. More...
2. Select the **Use filters** check box if filters (by code and by range) should be used for exported points. More...
3. If more fields are displayed for some file formats, complete them if required:
   - Configure **ASCII file properties** for a text format. By default, the created ASCII file will contain the type of the attributes, and all text values in this file will be written with quotation marks around them.
   - Click the **Code Style** button to setup the code style. More...

- Click the **Settings** button to configure the point style. More...
- Set a number starting at which **Alphanumeric points will be renumbered** for the formats which do not allow alphanumerical point names. By default, MAGNET Field Site will continue numbering from the last existing number of the job point.
- To **Store description as attribute**, select this check box for ESRI Shape format.
- To **Append points to existing file**, select this check box for CMM format.

Lines to File

Configure settings to export lines from the current job to a file:

1. By default the **Export areas as lines** check box is selected to save areas as lines in the created file.
2. If more fields are displayed for some file formats, complete them if required:
   - Configure **ASCII file properties** for a text format. By default, the created ASCII file will contain the type of the attributes, and all text values in this file will be written with quotation marks around them.
   - Click the **Code Style** button to setup the code style. More...
   - Click the **Settings** button to configure the line style. More...
   - Set a number starting at which **Alphanumeric points will be renumbered** for the formats which do not allow alphanumerical point names. By default, MAGNET Field Site will continue numbering from the last number of the job point.
   - To **Store description as attribute**, select this check box.

Areas to File

When you export areas from the current job to the ESRI Shape format file, you can **Store description as attribute**. To enable this option, select the check box.

Point Lists to File

Configure settings to export point lists from the current job to a file:

1. Select the **Select types of the points** check box if not all types of points in the point list should be exported. More...
2. Select the **Use filters** check box if filters (by code and by range) should be used for exported points. More...
3. If more fields are displayed for some file formats, complete them if required:
   - Configure **ASCII file properties** for a text format. By default, the created ASCII file will contain the type of the attributes, and all text values in this file will be written with quotation marks around them.
   - Click the **Code Style** button to setup the code style. More...
   - Click the **Settings** button to configure the point style. More...

- Set a number starting at which **Alphanumeric points will be renumbered** for the formats which do not allow alphanumerical point names. By default, MAGNET Field Site will continue numbering from the last existing number of the job point.
- To **Store description as attribute**, select this check box for ESRI Shape format.

Raw Data

Configure settings to export raw data from the current job to a file:

1. The **Export TS raw data** check box is selected by default if the format of the created file is intended for raw data collected with total stations.
2. The **Export GPS raw data** check box is selected by default if the format of the created file is intended for raw data collected with GNSS receivers.
   Note: Code, Attributes, String, Note, Control Code, Photo Name and Control flag are also exported with GPS and TS raw data into LandXML.
3. If more fields are displayed for some file formats, complete them if required:
   - Optionally, you can **Select additional point types** to write into the created file. More...
   - Select the **GPS points as SP** check box to save GPS points as manually entered points (Store Points) in the Carlson SurvCE RW5 and Trimble TDS RAW files.
   - Select the **Control codes as notes** check box to include control codes as notes into a TDS file. In this case you can make the TDS file compatible with the FBK format. To do this, select the **FBK compatible** check box; the **Alphanumeric points will be renumbered** field appears to set the first number that will be used to rename points. By default, MAGNET Field Site will continue numbering from the last existing number of the job point.
   - Optionally, you can **Rename alphanumeric points** for Topcon FC-6/GTS-7 and Topcon GTS-7+ files. To do this, select the corresponding check box; the **Alphanumeric points will be renumbered** field appears to set the first number that will be used to rename points.
   - The **Attributes** and **Attribute Names** check boxes are selected by default to write these values into the created Field Book file.
   - Set a number starting at which **Alphanumeric points will be renumbered** for the Field Book format which does not allow alphanumerical point names.
   - Configure **ASCII file properties**. By default, the created Topcon Custom GPS file will contain the type of the attributes.
   - Click the **Code Style** button to setup the code style. More...

Roads to File

When you export roads from the current job to:

- Topcon 3DMC Project file, MAGNET Field Site will rename alphanumeric point names. By default, it will continue numbering from the last existing number of the job point.
- LandXML file, the road string sets are also included. Optionally, you can store x-sections as Zones if you select the **Use Zone elements for x-sections** check box.

Localization to File

When you export localization data from the current job:

- To Topcon 3DMC Project file, MAGNET Field Site will rename alphanumeric point names. By default, it will continue numbering from the last existing number of the job point.
- File formats allow only WGS84 -> Local type of localization (for details, see) for export. Control Points are exported together with Localization data.

Surfaces to File

When you export surfaces from the current job to Topcon 3DMC Project file, MAGNET Field Site will rename alphanumeric point names. By default, it will continue numbering from the last existing number of the job point.

Multiple Data to File

To export muliple data from the current job to a file:

- Select Data from available for the selected format.
- Select Point filters to limit the number of the exported points.

When you export multiple data, the fields specific for every included data type will display. Complete them if required.

Data Selection

This dialog lists the **Available data** to export depending on the selected format.

- To select desired data types to export, check mark the boxes near the appropriate data types and click *Next*.
- The control next to this selection lists the number of the corresponding objects in the job.

Point Selection

Select the filters for the points to export from the drop-down list: All Points, By Point List(s), By Type(s), By Range and Code(s), By Type(s), Range and Code(s), or None.

If more than one filter is selected, they will be processed together and only points that meet the settings of all the filters will be exported. Select *None* you do not want to filter.

File Units

This dialog allows you to select units for the data being exchanged.

This option is available for all data types and formats containing distance/angular values. Depending on the data and format selected, you can set only distance, or distance and angle units if available.

Code Style

This dialog enables you to setup a code style, code information on a point, in the exported file.

1. To use an existing style, select it from the **Style** list and click  .
2. To delete any code style from the list, select it and click the **Delete** button.
3. To edit a code style:

- Select the style from the list.
- The **Available** field will display items you can add to a default set in the *Order* field.
- Use the arrows to move the necessary items between the fields and to arrange the items in the desired order.
- Click the **Separators** button to set code style separators. More...
- Click the **Control Codes** button to set names More...
- Click **Save** to save the changes in the code style.

Separators

This dialog lists the current separators for the selected code style and allows you to edit them.

- In **General Prefix** field you can enter a character/symbol to separate the code information from the rest of information on a point in the exported file.
- If required, enter other separators to delimit the items in the code style.

Control Codes

This dialog lists the control codes to create lines. The values can be replaced by user-defined ones during an export routine. The changes are applied only for the selected style. By default, MAGNET Field Site uses AS, AE, C, R as a linework package.

DXF/DWG Settings for Export

On this dialog you can select either the Point Style or the Line Style depending on the Data type selected to export to the file.

1. Choose a **Point Style** from the available styles:
   - *AutoCAD Points*: only point coordinates.
   - *AutoCAD Points with Text Fields*: point coordinates with text fields for point name, elevation, codes and attributes.
   - *Civil 3D/Land Desktop Point Objects*: Civil 3D points will be used.
   - *Carlson Point Blocks*: Carslon point blocks will be used. They include points and descriptions for them as block references.
   - *TopSURV Point Blocks*: TopSURV style will be used. It includes point name, elevation, codes, strings and attributes. displays points with the names as block references.
2. Choose a **Line Style** from the available styles:
   - *AutoCAD Lines*: only line coordinates.
   - *AutoCAD Lines with Text Fields*: line coordinates with text fields for line name, elevations, codes and attributes.
   - *TopSURV Line Blocks*: TopSURV style will be used. It includes line name, elevations, codes, strings and attributes.
3. The **Use 3D coordinates** check box is selected by default to export elevations with plain coordinates as well.
4. Select the **Use text font height** check box to open a field to manually/automatically set the height of text fonts to show the text (in CAD units). By default, it is auto.

Text File Format Settings

To configure settings of the format for the Text File:

1. Select the **Delimiter** symbol to separate data in the exported file. It can be a space, a comma, tabs or other symbol selected from the drop-down list.
2. Select the **Header in First Row** check box to output a header in the file.

3. In the **File Style** field:
    - From the drop-down list, select the order of fields in the format.
    - Click **Add** to create a new format style. More...
    - Click **Edit** to change an existing format style. More...
    - To delete an existing format style, click the **Delete** button.

Custom Style

To create a new style for Custom Formats:

1. Highlight the necessary items in the **Available** field or the **Order** field.
2. Use the appropriate arrows to move the necessary items between the fields. You can use only one type of code information in the file.
3. Use the appropriate arrows to arrange the items in the Order field in the desired order.
4. Click  to save the File Style and return to the Text File Format dialog. A new string appears in the File Style drop-down list.

Coordinate System

This dialog displays information about the coordinate system in the job whose data is exported. Select the **Coordinate Type** for the data in the text custom file exported.

Find out more on Coordinate Systems.

Units Format

This dialog enables you to select the desired format to represent data being imported from/exported to the file.

- From **Format (Lat/Lon)** select the format for degrees in Latitude and Longitude.
- From **Plane coordinates precision** select fractal length (precision) for plane coordinates from 0.0 to 0.00000000.
- From **Elevation precision** select fractal length (precision) for height from 0.0 to 0.00000000.

Import From File

To import data from a file to the current job:

1. Select the data type and the file format from which you will import data as required. More...
2. Select settings for importing points. More...
3. If required, select file units for imported data. More...
4. Select the file from which data will be imported.
5. Define settings for custom text formats. More...
6. Set up the coordinate system and coordinate type for imported data.
7. View the import status. More...

Import Data From File

To import different data types from files of either predefined or custom formats:

1. Select the **Data** type to import from the file.
2. Select the **Format** of the file being imported for the data type you selected. Each format can store a certain data group.
3. The **Select file units** check box is selected by default to configure units for values in the file. More....
4. Select the **Use Legacy Localization** check box when you import a localization file, which was created by all versions of Topcon Tools, TopSURV, Pocket 3D, 3D-Office and versions 1.* of MAGNET Tools and MAGNET Field. If you click the Use Legacy Localization check box, the Legacy mode is automatically set for horizontal localization in the pop-up menu of Localization.
5. The **X-sections include catch points** check box is selected by default to import x-sections that include catch points in some formats.
6. Click the **Settings** button if available to configure settings for some data type and formats. More...

   Settings For Text Custom Formats
   Import of Multiple Data Types

Settings for Import

Configure settings for some data type and formats:

1. Select the type of the points you import:
   - *Control Points*: the points with coordinates, known from the catalog. These are used for localization.
   - *Design Points*: points used as targets for staking.

2. Select the **Load as background** check box to import data from the file as a background image. If the box is unchecked, all data from the file will be stored as a set of the points/lines within a job. This will not happen if import of point lists has been selected.
3. Select the **Import block base points** check box to import block base coordinates as points.

Import of Multiple Data Types

After loading data from the file you can select the data types you want to import.

List of Imported Objects

This dialog shows the list with the objects loaded from the file. Select the check boxes before the appropriate objects to import.

Wrong Objects

The dialog is being shown when an object with the same name as the imported one already exists in the job. Depending on the object type, the following options are available:

1. Select **Overwrite?** to replace the object in the database with the imported one.
2. Select **Rename?** to give a new name to the imported object in the Start Name field.
3. Select **Prefix?** to add the entered prefix value to the imported object name.
4. Select **Suffix?** to add the entered suffix value to the imported object name.
5. Click **Yes** to accept the decision.
6. Click **Yes To All** to accept the same decision for all similar cases.
7. Click **Skip** to skip the object without importing.
8. Click **Skip All** to skip all the objects with names that coincide with the names of existing objects, without importing.

Export To 3DMC

The To 3DMC dialog allows you to export data to 3DMC file and SiteLINK 3D users.

Select the required option to Export job to:

1. **To 3DMC File (.TP3)**. More...
2. **SiteLINK 3D user (.TP3)**. More...

Also, you can *Transfer other file(s) to* **SiteLINK 3D user**. More...

To 3DMC

To export job to 3DMC File for SiteLINK 3D user:

1. Select a user group from the **Group** drop-down list: All, Hardware, Software, Machines.
2. Select **Online only** to show only online contacts.
3. Select contacts from the table of the available user *Names* and their *Types*.
4. Click **Next** to proceed with export to MC project. More...

To MC project

To export *Multiple* data from the current MAGNET Field Site job to *Topcon 3DMC project*:

1. **Data** and **Format** fields cannot be changed.
2. The **Select file units** check box is selected by default that allows you to view units which are set by default for the data exported.
3. Select the **Export areas as lines** check box to export area data as line data to the MC project.
4. The **Alphanumeric points will be renumbered starting at**: field displays a number at which MAGNET Field Site will start renaming points with alphanumeric names as the Topcon 3DMC project demands points that have only numerical names. MAGNET Field Site will continue numbering from the last existing number of the job point.
5. Click the **Code Style** button to setup the code style. More...
6. Click **Next** to start the export.

SiteLINK 3D file

To transfer files other than 3DMC file to SiteLINK 3D users:

1. Select a user group from the **Group** drop-down list: All, Hardware, Software, Machines.
2. Select **Online only** to show only online contacts.
3. Select contacts from the table of the available user *Names* and their *Types*.
4. Click **Add** to select a file to be added to the list.
5. If necessary, you can remove *All* or *Selected* file. Select the corresponding option from **Remove**.
6. Highlight the required file and click **Send**.
7. **Transfer status** will show the export in progress.

Import From 3DMC

The import of data from a 3DMC project is a part of general import process that is copied into a standalone tool.

To import *Multiple* data from the *Topcon 3DMC project* to the current MAGNET Field Site job.

1. **Data** and **Format** fields cannot be changed.
2. The **Select file units** check box is selected by default that allows you to view units which are set by default for the data imported.
3. Click the **Settings** button to configure additional settings. More...

Enterprise Upload

To upload data from the current job to an enterprise project:

1. Select a **Project** from a list of available projects. After you select the project, the names and types of the inboxes appear in the field. Check mark the desired inbox.
2. Click **Add** to upload a file. The name and path to the file will be displayed in the corresponding field.
3. If required, click **Clear All** to erase selections.
4. Click **Upload** to start uploading data to the selected inbox. *Upload status* will show the upload progress and the result.

Enterprise Download

To download uploaded data from an enterprise project to your device:

1. Select a **Project** from a list of available projects. After you select the project, the names and types of available inboxes appear.
2. Select the inbox that contains the needed files. The names of the uploaded files and the dates of uploading appear. Select the file(s) to be downloaded.
3. If required, click **Refresh** to renew the content of the selected inbox.
4. Click **Download** to start downloading data to your device. *Download status* will show the download progress and the result.

Chat

Click an icon for text messaging. The *Chat* dialog contains three tabs:

The Users tab displays a list of the names of all MAGNET Enterprise users who are currently assigned to a chat.

1. To make the list show only the users with online presence status, check the **Online** box. You will see the users who are ready for immediate connection.
2. To start chatting:

- Select the user you want to chat with by checking the box near the name.
- Click the **Start Chat** button to initiate conversation.
- Type in the massage you want to post to the selected user.

The SiteLINK 3D tab displays a list of the names of all SiteLINK 3D users who are currently assigned to a chat.

1. Select a user group from the **Group** drop-down list: All, Hardware, Software, Machines.
2. Select **Online only** to show only online contacts.
3. Select contacts from the table of the available user *Names* and their *Types*.
4. Click the **Start Chat** button to initiate conversation.

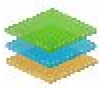
The Chats tab displays a list of the users with whom you have already started chatting.

Edit folder

Click an icon to perform the task:

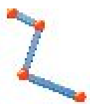
Points

Edits properties of existing points and add new points manually.



Layers

Edits layer properties and add new layers manually.



Polyline

Edits the polyline properties and create new polylines manually.

Raw Data

Edits raw data and re-compute coordinates.

Surfaces

Edits surfaces.

Roads

Opens a subfolder with six options to edit the road properties and design new roads.

Stake Reports

Edits stake reports.

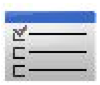
Codes

Edits code properties and add new codes.

 Edit Points

The Points:<Coordinate Type> dialog contains the list of the points stored in the application database. The list provides you with the basic information about every point: a point's type and name, coordinates, codes and note. The title of the dialog shows the type of the coordinates displayed. The icon near the point name indicates the point type that is the method the point is determined. Learn more about the icon descriptions for the points.

The buttons available in this dialog serve the following purposes:

- Expand the **Find** drop-down list and select an option to find a point:
  By Range,
  By Code,
  By Code String,
  By Radius,
  By Name or
  By Layer.
- Click **Find Next** to find the next point in the list that satisfies the same conditions as the previous Find.
- Click **Delete** to delete the point from the list.
- Click **Edit** to open the Edit Point dialog
- Click **Add** to create a new point.
- Click  to configure the display settings. More...

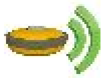
The icon  brings up the pop-up menu of additional options. More...

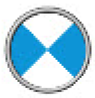
Point Icon Descriptions

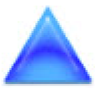
 GPS stationary (topo)

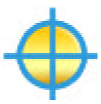
 offset topo

 GPS kinematic (auto topo)

 RTK base

 sideshot

 control

 design or imported

 staked

 calculated

 entered manually

---

Points pop-up menu

If required, you can switch on any of the following functions:

- Select **Setup Columns** to customize the arrangement of data columns in the list of points. More...
- Select **PTL Mode** to switch on the PTL (Point-To-Line) Mode. The From Point and To Point columns will appear in the list to show the reference line.
- Select **Show Scan Points** to display the scan points in the list.
- Select **Show AutoTopo Points** to display the AutoTopo points in the list.
- Select **Recompute** to recompute the point coordinates after editing the point's raw data (if the recomputation was not performed in Raw Data).

Setup Columns

To customize the arrangement of columns in the list of points:

1. Highlight the necessary items in the **Available** field or the **Selected** field.
2. Use the appropriate arrows to move the necessary items between the fields.
3. Use the appropriate arrows to arrange the items in the Selected field in the desired order.
4. Click to save the selected order of columns and return to the Points dialog.

Select Points By Range

To select points involved in a specified range:

1. In the **Range of Points** field, specify the names of the points to find. These can be specified by a range or by enumeration individually. The symbols ';', '.' or ',' can be used for *Name Separator* and '-' for *Range Separator*.
2. Click to save the settings and return to the Points dialog with the first found point highlighted. To find the next point in the range, click **Find Next**.

Select Points by Code

All the codes existing in the current job and their descriptions will be listed in a table.

1. Select the code(s) for which you want to find all the points having that code:
   - Selection can be done by placing check marks in the list, against the desired codes.
   - Click **Check** and **Uncheck** to toggle the highlighted item(s) on and off, respectively.
   - Click **Select All** to choose all the items at a time.
2. Click to save the selections and return to the Points dialog with the first found point highlighted. To find the next point with the selected code(s), click Find Next.

Select Points by Radius

To find the points which reside in a circle:

1. Select the **Point** either by entering it manually or by selecting it from the map or list . This will be the center of the circle.
2. Enter the **Radius** (distance) around the selected point in the current units.
3. Click to save the settings and return to the Points dialog with the found points highlighted in the list.

Select Points by Code Strings

To select the points by Code Strings:

1. Select the **Code** from the drop-down list of existing codes which have strings.
2. Select the **String**(s) you would like to use for the code string combination.
3. Click to save the selections and return to the Points dialog with the first found point highlighted. To find the next point with the selected code string(s), click **Find Next**.

Find Point By Name

To find a point by its name:

1. Enter the name or part of the name of the **Point**.
2. Select the **Match entire name** radio button if the whole name was entered in the Point field.
3. Select the **Match partial name** radio button if a part of the searched name was entered in the Point field.
4. Click to save the settings and return to the Points dialog. If found, the point will be highlighted in the list.

Select Points By Layer

The Layers dialog enables you to select points which are on the same layer. To do this:

1. Select the layer from the list of existing layers as required.
2. Click to save the selection and return to the Points dialog. When found, the points will be displayed.

Edit Point

This dialog can contain the following tabs, depending on the properties of the highlighted point:

- Point tab
- Layer/Style tab
- Cut Sheet tab
- Check Points tab
- WA tab
- PTL tab
- Image tab

Point tab

You can edit /view:

1. Name of the **Point**.
2. Code and attribute information for the point.
   - You can select a **Code** from the drop-down list. Code needs to be defined at the time it is entered if it is not a code that exists in the Codes dialog.
   - If the code type is Line or Area, an icon will display that the point belongs to a line. Set a string and, if required, a control code.
   - Press the **Code** button to view information on the Point Attributes dialog. You can set two control codes, and attribute values for the code.
   - Enter any additional information about the point in the **Note** field.
3. The display of coordinates depends on the selected coordinate system.
4. If required, select the **Control Point** check box to add a new point as a control point for localization.
5. Click  to save the point. If a point exists and you try to save another point with the same name, a dialog displays a prompt that the point already exists. More...

String

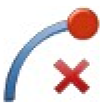
String is a specifying parameter for a code that allows you to group the objects with one code according to some specified attribute. For example, the code "Pole" also has the "Jones'" string. Processing the points, you will be able to select only the poles of Jones' and will not take into consideration all the other poles.

Control Codes

Additional manipulations of polyline can be performed using control codes for the points with the same code-string combination. Up to two control codes can be specified for every code associated with a point to store the points which will be connected to form open or closed polylines. Select the codes of the Line type to use for such points.

The supported control codes of line behavior:

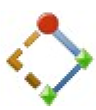
Arc Start(AS)
Indicates the start of an arc. Arc parameters are determined by the presence of additional points in the line. These points can create the line segment with the arc start point which will act as the tangent to the arc.



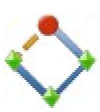
Arc End(AE)
Indicates the end of an arc. Arc parameters are determined by the presence of additional points in the line. These points can create the line segment with the arc end point which will act as the tangent to the arc.

Circle Edge(CE)
When this code is applied to the first point of a three-point polyline, this point and next two points define the edge of the circle.

Circle Radius (CR)
When this code is applied to the first point of a two-point polyline, it indicates the center of a circle. The second point will define the radius of the circle.

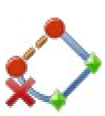


Rectangle(R)
When this control code is applied to the third point of a three-point polyline, this results in the automatic creation of a fourth point of a parallelogram whose diagonal is specified by the first and the third point.



Close (C)
When this code is applied to a point, it closes the polyline with each next point.

Arc End & Start (AE&S)
When this dual control code is applied to a point, it indicates the end of one arc and the start of another arc.



Arc End & Close(AE&C)
When this dual control code is applied to a point, it indicates the end of the arc and creates the line segment with the arc start point.

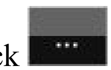
Note: If only one point is between the arc start and end points, the arc is formed such that all the three points lie on the arc. If there are two, or more than two points, between the points with the AS and AE control codes, the points are all connected by straight line segments.

Note: MAGNET Field Site will not use this linework package if the *Allow Custom Control Code* box on the Global screen is selected. In this case, the user can enter any string to mark it as a control code. MAGNET Field Site will not interpret these control codes.

Layer/Style tab

By default the layer and point style are defined By Code for the selected point.

To edit the Layer/Style information:

1. Define the **Layer** in which to have the point. Select it from the drop-down list. Click  to edit the layers. More...
2. Define the **Point Style** for drawing. Select the point type and color to be used with the point.

Cut Sheet tab

This tab is present if the selected point is staked and stored.

The list displays the Name of the staked point, the dE, dN, and dH vector of the staked point from the design point, the Coordinates of the staked point and any Notes associated with the staked point.

Check Points tab

This tab is present if the point has any check points associated with it.

The list displays the Name of the check point, the dE, dN, and dH vector of the check point from the recorded point, the Coordinates of the check point and any Notes associated with the check point.

WA tab

The WA tab displays a list of stations that can be used in the Weighted Average for the point. Also, the E, N, U residuals, WA Control (whether its used or not), and notes are displayed.

Select a station in the list. When you click on the **Use in WA/Exclude from WA** button, the appropriate action will be taken: the station is added for the WA or excluded from the WA.

PTL tab

This tab is displayed when the selected point is a PTL point or when the PTL mode is selected when adding a point.

1. Select the **Start** and **End Reference Points** to define the line for the PTL offsets. These points can be selected from the map or from the list of job points by clicking on or , respectively.
2. Set **PTL Offsets** in the current distance units:
   - In **Line**, set the distance along the line from the Start Reference Point.
   - In **Offset**, set the distance in a direction perpendicular to the line.
3. **Height** is the height of the current point.

Note: In this tab you cannot edit any field for the measured PTL point.

Image/Photo tab

This displays a photo note attached to the point.

To edit the photo note:

- To add an image for the point, you can use:

  either to select a photo from existing ones

  or to capture an image from the controller's camera, if the controller supports the camera. More...
- Use the << and >> buttons to scroll through the photos attached to the point.
- Click if required to remove the photo note currently displayed for the point.
- If required, use the following icons to geotag the image:

   to show the current data and time in the photo note.

   to show coordinates of the point.

   to display compass readings taken at the time you take the photo note for the device that supports the camera and the compass.
  The Compass Calibration dialog will prompt you to calibrate the compass before taking a photo (if not

already calibrated). The Compass Calibration dialog appears automatically only once. If you skip the calibration, you will not be prompted again as long as you edit the point. You will be able to calibrate the compass any time by clicking the *Calibrate Compass* option from the pop-up menu .

Compass Calibration

The Compass Calibration dialog allows you to process the compass calibration. The application provides a graphic image describing calibration movement.

- If you do not want to process the calibration, click the **Skip** button to ignore the compass calibration, and no direction will be provided even if the *Show Direction* is selected.
  Note: If you skip the calibration, you will not be prompted again as long as you edit the point. You can calibrate the compass any time by clicking the *Calibrate Compass* option enabled in the pop up menu.
- Click the **Start** button to start calibration. The calibration time depends on the device used. For example, it is 10 seconds for the FC-236, and 60 seconds for the GRS-1 device.

Image Capture

The Image Capture dialog allows you to take a photo note for the point.

You can turn on or turn off **Show Coordinates** and **Show Direction** from the Settings dialog that displays by clicking the Settings button in the top right corner of the dialog. More...

Camera Settings

The dialog allows you to geotag every image when you capture it in the current job.

- Select **Add Timestamp** to display the current date and time over the photos.
- Select **Add Coordinates** to display the coordinate system and the coordinates of this point over the photos.
  Note: If you select *Add Coordinates* and the WGS84 location of the point is available, the coordinates will be displayed in DDMMSSSS format; otherwise, no coordinates will be displayed or saved in the image.
- Select **Add Direction** to display compass readings taken at the time you capture the image. This option is enabled if the device supports the camera and the compass.
  Note: The direction is displayed over the photos provided the compass has been calibrated.

Point Attributes

The dialog allows you to set attributes for the point being created or edited in the following tabs:

- Code Attributes tab
- Layer
- Photo
- Note

Code Attributes tab

To set code attributes for the point:

1. Select a **Code** from the drop-down list. The Code list shows all the codes to use in the job.
2. If required, select associated two **Control Codes** from the drop-down lists. The control code is a special type of code that can be used by some graphic tools for the interpretation of the survey results.
3. Select a string for the code of the line or area type.
4. The attribute table lists all available attributes for the code and allows you to enter/select its value.
    - Click the **Props** button to open the Attribute Ranges dialog, which displays the valid ranges for the attributes for the selected code. More...
    - Click **Repeat** to set the previous saved value.
    - Click **Default** to set the default value.
5. To add **Multiple Codes**, click on this button. More...

Attribute Ranges

The selected **Code** name is displayed.

The table displays a list of attributes associated with the code. When you highlight a row, the title of the properties column changes to indicate what is being displayed. For example, for Text type of attribute it would say Max Chars.

Multiple Codes

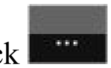
The Multi-Code displays a list of all codes, strings and attributes for the point being edited.

- Click **Add** to add another code to the list. More...
- Click **Edit** to change the code selected from the list. More...
- Click **Delete** to delete a selected code from the list.

Layer/Style tab

By default the layer and point style are defined By Code for the selected point.

To edit the Layer/Style information:

1. Define the **Layer** in which to have the point. Select it from the drop-down list. Click  to edit the layers. More...
2. Define the **Point Style** for drawing. Select the point type and color to be used with the point.

Point Check

If a point exists and you try to save another point with the same name, a dialog displays a prompt that the point already exists.

1. Displays the offsets of the point you are trying to save from the point that is saved.
2. This dialog enables you to:
   - **Overwrite** on the existing point
   - **Rename** the point to store as another point
   - or **Store as check point** to the existing point
3. If you select to store the point as a check point, you can enable using it in weighted averaging positions. To do this, check **Use in weighted average**. More...
4. In RTK surveys, if checked, the **Correct base** option allows you to setup the Base at unknown position. In this case, you can correct the base position if an observed Topo point has known coordinates stored in the job and has the same name as the base. The known coordinates of the observed point are not replaced with the observed coordinates and are used to correct the Base coordinates.
   After either closing the Topo dialog or moving to another tab, recomputations are performed, and the coordinates of all points are updated using the new Base coordinates.
   Note: For *Correct Base* to work properly, the coordinate type selected in Display must be the same as for the known coordinates of the observed Topo point.

Image / Photo

This displays a photo note attached to the point.

To edit the photo note:

- Use the << and >> buttons to scroll through the photos attached to the point.
- Click **Add** to attach an image to the note.
  Note: If the controller supports the camera, there will be two options available. Click either **Browse** to select a photo from existing ones or **Capture** to capture an image using the controller.

- Click **Edit** to delete the currently selected photo and attach a different one instead.
  Note: If the controller supports the camera, there will be two options available. Click either **Browse** to select a photo from existing ones or **Capture** to capture an image using the controller.
- Click **Delete** to remove the photo note currently displayed for the point.

The icon  brings up the pop-up menu that contains three additional options for a photo note. More...

Photo Note Pop up menu

- If required, click **Show Coordinates** to display the coordinate system and the coordinates of this point over the photos.
  Note: If you select *Show Coordinates* and the WGS84 location of the point is available, the coordinates will be displayed in DDMMSSSS format; otherwise, no coordinates will be displayed or saved in the image.
- If required, click **Show Direction** to display compass readings taken at the time you take the photo note. This option is enabled for the device that supports the camera and the compass.
  Note: The direction is displayed over the photos provided the compass has been calibrated.
- When you first open the Photo Note tab and select the *Show Direction* option, the Compass Calibration dialog will prompt you to calibrate the compass before taking a photo (if not already calibrated). The Compass Calibration dialog appears automatically only once. If you skip the calibration, you will not be prompted again as long as you edit the point. You will be able to calibrate the compass any time by clicking the *Calibrate Compass* option enabled in the pop up menu.

Note: When you first open the job, the *Show Coordinates* and *Show Direction* options are unchecked. After you select one or both options for one photo, the application will keep the selections. The selections will be lost when the device is shut down. Also, you can turn on or off *Show Coordinates* and *Show Direction* from the Settings dialog in the Image Capture dialog.

Note

The Note dialog is used to enter additional information. Type the text in the Note field.

Edit Codes

The Code-Attributes dialog contains a list of **Codes** used for the survey, the list of **Attributes** for each code, and a set of tools for editing:

1. Click **Add** to create a new Code or Attributes (depending upon the list to which the button belongs).
2. Click **Edit** to change the properties of the highlighted entry (depending upon the list to which the button belongs): Code or Attributes.
3. Click **Delete** to delete the highlighted entry from the list.

Code

To create a new code or edit an existing code, enter the code details:

1. The **Name** of the code.
2. The **Description** for the code.
3. The **Type** of the objects that the code describes: *Point*, *Line* or *Area*. Each type has an associated plotting style(s) that can be edited.
   - *Point*: Set a symbol and attributes for points and color. Can be used in *Surface*.
   - *Line:* Set a symbol for nodes, style and thickness for lines, and color. Can be used in *Surface* and as a *Breakline*.
   - *Area:* Set a symbol for nodes, style and thickness for the boundary, filling style and transparency for the area, and color. Can be used in *Surface*, as a *Breakline* or/and an *Exclusion Area*.
   - *Attribute:* Select the radio buttons on this tab to enter the code at the beginning of a line/area (*Start*) or at every node point along the line/area (*Each Node*)
   - *Surface:* Select the corresponding check box on this tab to use the code in *Surface*, as a *Breakline* or/and an *Exclusion Area*
4. Determine a **Layer** for the code if required.

Attributes

To create a new attribute or edit an existing attribute, enter the attribute details:

1. The **Name** of the attribute.
2. The **Type** of the attribute. *Text*, *Bool, Date-Time, Integer, Menu* or *Real Number*. Each type has an associated settings that can be edited.

- *Text:* Select this if the attribute value is an alpha-numeric string. Set the maximum number of characters you can enter.
- *Bool:* Select a boolean value.
- *Date-Time:* Use the default (current date) and time for the code attribute or set the date and time from the drop-down list.
- *Integer:* Select this if the attribute value is an integer. Set the minimum and the maximum values of the attribute.
- *Menu:* Select this to assign a list of values to select for the attribute. The values are entered in the field and added to the list with the Add button. To remove a value from the list, select the value from the list and click .
- *Real Number:* Select this if the attribute value is a real number. Set the minimum and maximum values of the attribute.

3. The **Default** value for the code attribute. You can leave this field empty only when the *Required* check box is not selected. Enter the value and select the *Required* check box to make sure that the default value for the code attribute is defined.

Edit Linework

The Polyline dialog contains a list of existing Polylines, and the two windows, that present the general view of the selected polyline in the horizontal and vertical planes. To view the current selected polyline in a larger map, double-click one of the map plots.
Note that the double click on the map and property buttons are disabled if accessing this dialog from the Alignment Pair dialog.

- Click **Edit** to edit the properties of the selected polyline:
  Points in Line
  Layer/Style
- Click **Add** to create a new polyline.
- Click **Delete** to delete the selected line from the list.

Points in Line

The Points in Line tab allows you to edit the selected line. The list contains the points with their codes that compose the line, and the plot displays the line image.

To edit the line:

1. If required, change the **Name** of the line.
2. To delete a point from the line, highlight the point and click the red minus button.
3. To obtain information about a single point, highlight the point and click the Info button.
4. The icon next the Select Points field shows the control code from the linework package for the highlighted point. Click the icon and, if required, select another control code. Also, the selection of points *From Map* enables you to set such a control code.
5. From the *S*e**lect Points** drop-down list, choose an option to select a point(s) from the job to add to the end of the line.
   - *All* - all the job points.
   - *By Range* - points from a range. More...
   - *By Code* - points of a selected code. More...
   - *By Code String* - points of a selected code string. More...
   - *By Radius* - points around a selected point at a certain distance. More...
   - *By Name* - a point found by its name. More...
   - *By Layer* - points on a selected layer. More...
   - *From Map* - points selected graphically from the map. Select the points by clicking them on the map;

points that are sequentially clicked are connected with a line. More...

- *From List* - points from a list of points. More...

6. Use the up and down arrows to move the highlighted point up or down in the ordering.

Learn more about the icon description on the dialog.

Select Points From List

Lists the points in the current job to select.

The buttons available in this dialog serve the following purposes:

- Expand the **Find** drop-down list and select an option to find a point:
  By Range,
  By Code,
  By Code String,
  By Radius,
  By Name,
  By Layer or
  From Map
- Click **Find Next** to find the next point in the list that satisfies the same conditions as the previous Find.

The icon  brings up the pop-up menu of additional options. More...

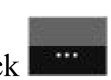
From List pop-up menu

If required, you can switch on any of the following functions:

- Select **Show Scan Points** to display the scan points in the list.
- Select **Show AutoTopo Points** to display the AutoTopo points in the list.
- Select **Edit Points** to edit a point in the job. More...

Layer/Style

To edit the linework properties to display the line and existing points in the selected linework on the map:

1. Select the **Layer** from the drop-down list of existing layers in the job. Click  to edit a layer.
2. In **Point Style**, select an icon for the points from the drop-down list and check its display.

3. In **Line Style**, select a style for the lines and the **Width** of lines in points.
4. Click **Color** to choose the color for the point and the line.

Edit Raw Data

All the raw data collected is displayed in a list on this dialog. When a line is selected, the column titles reflect the data in that line.

The Raw Data list includes information about:

- *Name*: Point name and the icon displaying the type of the point. Learn more about the icon descriptions for the points. The icon stands for a raw note made in the field during the survey.
- *Type*: The type of measurement. *Reset GNSS* type stands for the action performed manually or automatically to reset the RTK engine.
- *Codes*: Codes for the point, also the Strings associated with the codes if the String display is selected.
- *HI/Ant Ht*: For the Optical mode - the height of the instrument; for the GPS+ mode - the antenna height.
- *Coordinates*: The Coordinates of the Base are displayed and the vectors to the base from the rover of the collected points. The vectors are only displayed for the points collected with fixed solutions.The TS point coordinates.
- *Ctrl Code*: Control codes associated with the point.
- *Solution Type, PDOP, H/V RMS, number of GPS and GLONASS Satellites:*For points collected in the GPS mode.
- *Notes*: The short note for the point.
- *Local Time*: It is the controller date and time when the point was stored. For points collected in the GPS mode, it is the time of the epoch whose coordinates are stored or the time of the last accepted epoch when averaging.

The buttons available in this dialog serve the following purposes:

- Expand the **Find** drop-down list and select an option to find a point:
  By Code,
  By Name
- Click **Find Next** to find the next point in the list that satisfies the same conditions as the previous Find.
- Click **First / Last** to move the cursor to the first or last point.
- Click **Edit** or double click on a line in the list to see and edit the data in a separate dialog. You can edit:
  TS survey: the point name, code, height of reflector, backsight azimuth, note, height of instrument and scale.
  DL survey: the name, code, note and vertical offset for the side-shot point.
  GPS survey: the name, code, antenna height, antenna type, height measurement type, note and offset data for the surveyed point.
  GPS Base Station, started out of the current job: Phase Center coordinates and use of relative antenna calibration data.
  GPS Base Station, started in the current job: Mark coordinates, the point name and code, the antenna type,

height, height measurement type and use of relative calibration data.
Note: The editing does not result in immediate recomputations of the point coordinates. Instead the raw data may be downloaded onto office software and recomputations can be performed there.
- Click **Recompute** to re-compute the point coordinates after editing the point's raw data.

The icon  brings up the pop-up menu of additional options. More...

Raw Data pop-up menu

- **Job Info**: displays the Job Info dialog.
- **Show Raw GPS+/TS**: If the GPS+ mode is active, the GPS+ raw data is displayed by default. If you want to display the TS raw data in the list, select this menu. Similarly, if you are using the Optical mode, the TS raw data is displayed and optionally the GPS+ raw data.

Edit Optical Raw Data

The Edit Raw Data dialog displays the properties of the selected record and allows you to change the name, code, notes, and some other record specific data (for instance, the height of instrument/reflector for total station survey and vertical offset for leveling).

Edit GPS Raw Data

The Edit Raw Data dialog displays the properties of the selected record and allows you to change the name, code, notes, and some other record specific data (for instance, GPS antenna information and offset data).

Edit Raw Base Station

The GPS base station can be started at a point (mark) of known coordinates in the current job or run out of the job and may show in raw data the Mark or Phase Center Coordinates, respectively.

The point's name, code, mark coordinates and the antenna information can be changed for the base started in the job. For the base, which runs out of the job, you can edit the phase center coordinates and use of relative antenna calibration data.

Edit Surfaces

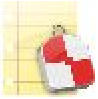
The Surfaces dialog contains a list of existing Surfaces, and a panel of the general view of the selected surface.

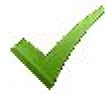
- Click **Edit** to edit the selected surface properties. More...
- Click **Delete** to remove the selected surface from the job.

Surfaces Properties

The Surface Properties dialog includes the Info tab which displays information about the area covered the selected surface and the Style tab that allows you to edit:

- The **Name** of the surface.
- The **Line Style**, the line's form, width and color.

 Edit Stake Reports

The Stake Reports List dialog displays a list of existing reports in the job and information about each report: its name, type (icon and name), configuration and references . The  icon near the report name means that the report is set as current for this type. Click the icon to change the report status.

The buttons on the dialog serve the following purposes:

- Click **View** to view the full report selected. More...
- Click **Delete** to remove the highlighted report from the list. You will have to confirm deletion twice before the report data is deleted.
- Click **Edit** to edit the report selected in the list. More...
- Click **Add** to create a new report.

Stake Report

To edit the selected stake report:

1. Enter the **Name** of the report.
2. Select the **Report Type** from the list.
3. If required, edit the report **Configuration** for the report type.
4. Select the **Set Current** check box to set a new report current.

View Stake Report

Displays design references and appropriate information for this report type.

Displays the Stakeout Information for the point being saved.

Edit Roads folder

Click an icon to perform the task:

Roads
Edits properties of existing roads and create new roads.

Horizontal
Edits an available horizontal alignment and create a new one.

Vertical
Edits an available vertical alignment and create a new one.

Templates
Edits properties of an existing x-section template and create a new one.

X-Sections
Edits available x-sections and create a new one.

String Set
Edits an existing set of strings and create a new set.

Edit Roads

The Roads dialog displays a list of the roads in the current job, and the plot of the selected road in the horizontal and vertical planes.

- Click **Add** to create a new Road. More...
- Click **Edit** to view/edit the parameters of the selected road. More...
- Click **Delete** to remove the selected road from the job.

The icon brings up the pop-up menu of additional options. More...

Roads pop-up menu

If required, you can exchange roads:

- Select *Import Road(s) From Job* to copy roads from another job. More...
- Select *Import Road(s) From File* to import roads data from a file. More...
- Select *Export Road(s) To Job* to copy roads to another job. More...
- Select *Export Road(s) To File* to create a file with the roads data. More...

Add/Edit Roads

In this dialog you can edit the properties of the road's Alignment and Surface.

Edit Road's Alignment

To edit the selected road alignment or to add a new road alignment:

1. Enter the **Name** of the alignment.
2. Select the **Layer** in which this road should be placed. The associated ... button brings up the dialog to edit Layers.
3. In the **Horizontal Alignment** drop-down list you'll find all available horizontal alignments. Select one for your Road. The associated ... button brings up the Horizontal Alignment list where you can add, edit and delete Horizontal Alignments.

4. In the **Vertical Alignment** drop-down list you'll find all available vertical alignments. Select one for your Road. The associated button brings up the Vertical Alignment list where you can add, edit and delete Vertical Alignments.
5. Enter the **Start Station** of the Road. Start Station is the start position along the Road Alignment.
6. Enter the **Interval** at which the next station position will be decided. Negative values can be entered, which will cause the advancement to decrease instead of increase along the line.

Edit Road's Surface

The road surface can be described through either *X-Section Set* or *String Set*.

- Toggling the button between the two will change the contents of the drop-down list. If you want to associate your road's surface with a cross section set or a string set, select one here. Click to edit the Cross Sec-tion Sets or the String Sets.
- In the **Working Corridor** field, enter:
  - *Left* side working corridor. The left corridor must be lower or equal to the right corridor.
  - *Right* side working corridor.

To calculate the road's points, click the icon and select Calculate Road Points from the pop-up menu.
More...

Calculate Road Points

To generate points along, to the right and to the left of the center line of the road, along its entire length:

1. In the **Points to Generate** field, define the points to generate:
   - Select the *Centerline Points* check box to generate the center line points.
   - Select the *Points Right of Centerline* check box to generate the points to the right of the center line.
   - Select the *Points Left of Centerline* check box to generate the points to the left of the center line.
   - Also, if it is desired to include the *Transition Points*, place the check mark in the corresponding field, select the types of transition points (More...) and select a *Prefix/Suffix* for them, if necessary, in the appearing field below.
2. Enter the **Station Interval** between the generated points. By default, it is the Station Interval set in the Start Pt tab in the Roads dialog.
3. Click the **Next** button to continue until the buton is available. Click it to open the *Calculating Road Points* status screen.

Transition Points

This dialog allows you to select the types of transition points to generate.

1. Select the corresponding check boxes to choose:
   - end point on the horizontal alignment
   - middle point on curve of the horizontal alignment
   - end point on the vertical alignment
   - high point on the vertical alignment
   - low point on the vertical alignment
2. Click  to save selection.

Centerline Points

To calculate points along the center line:

1. Enter the **Name** of the first point.
2. Select the code from the drop-down list. Click  to set the point attributes of the points being generated. More...
3. If required, select **Prefix/Suffix** to be added to the generated point name.
4. Select the **Save Points To Point List** check box if it is necessary to save the generated points to a separate points list. When checked, a field appears where the name for the list can be set.

Right/Left Offset Points

To calculate points to the right or left of the center line:

1. Enter the **Name** of the first point.
2. Select the code from the drop-down list. Click  to set the point attributes of the points being generated. More...
3. If required, select **Prefix/Suffix** to be added to the generated point name.
4. Select the **Save Points To Point List** check box if it is necessary to save the generated points to a separate points list. When checked, a field appears where the name for the list can be set.

5. Set the offset of the point from the center line along two dimensions: horizontal (the *Right/Left* field) and vertical (the *Up/Down* field) relative to the surface (the **Surface Offset** type) or to the horizontal line (the **Flat Offset** type).

Edit Horizontal Alignments

The dialog displays a list of the horizontal alignments in the current job, and the plot of the selected horizontal alignment.

- Click **Add** to create a new horizontal alignment.
- Click **Edit** to view/edit the currently selected horizontal alignment. More...
- Click **Delete** to remove the selected horizontal alignment from the job. You will be asked for confirmation.

Add/Edit Horizontal Alignments

In this dialog you can edit the elements of the Horizontal Alignment and its Start Point.

The icon brings up the pop-up menu. Select *Edit Points* to edit a point in the job. More...

Start Point

The Start Pt tab allows you to edit the following parameters:

- In **Alnt Name**, the name of the horizontal alignment.
- In **Start Pt**, the name of the start point that can be entered manually (if a new point name is entered, the point will be created with the coordinates entered in the North, East and Height fields) or chosen from the map or from the list .
- The point code can be entered manually or chosen from the drop-down list for a new point. Click to enter the point attributes.
- **North** and **East** display the local plane coordinates of the point.
- In **Start Sta**, enter the starting station for the horizontal alignment.

Horizontal Alignment

The Hz tab shows the list of horizontal alignment elements, the horizontal alignment plot and the start and end station (or chainage) of each element.

- The element list has the following columns:
  Element: the icon and the name of the element: line, spiral, curve, or intersection point
  Length: the length of the element

Azimuth: the azimuth at the beginning of the element;
Radius: the radius of the curve, spiral or intersection point (the radius of the spiral is the radius at the end of the 'incoming' spiral or at the beginning of the 'exiting' spiral; the radius of the intersection point is the radius of the corresponding curve.)
A1: spiral constant 1: Length of Tangent-Spiral to Spiral-Curve. The spiral constant is the square root of the product of the length and the radius of the spiral.
A2: spiral constant 2: Length of Curve-Spiral to Spiral-Tangent. The spiral constant is the square root of the product of the length and the radius of the spiral.

- Click **Add** to select elements to add after the last element (Line, Curve, Spiral, or Intersection Point).
- Click **Insert** to select elements to insert at the selected location in the list.
- Click **Edit** to change the properties of the selected element.
- Click **Delete** to remove the highlighted element from the horizontal alignment.
- Use the arrow icon to hide/show the plot.

Line

The Line dialog allows you to add a line:

- Enter the **Length** of the line element.
- The **Azimuth** is editable only for the starting element of the road because, by default, the azimuth is set tangent to the previous element.
- To change the azimuth of all other elements, clear the check mark from the **Tangent to Previous** box.
  Note*:* Use caution when setting the azimuth, since road elements are usually tangential to each other.
- The plot will show the element's appearance.

Curve

The Curve dialog allows you to add a curve:

- Enter the **Radius** of the curve or one of the two parameters unambiguously defining the radius: *Chord Angle* or *Curve Angle*.
- Enter the **Length** of the curve element or one of four parameters unambiguously defining the curve length: *Chord*, *Tangent*, *Middle Ordinate* (the distance from the midpoint of a chord to the midpoint of the corresponding curve), *Delta* (the angle between the radii corresponding to the curve), or *External* (a section of a line connecting the center of the arc with the vertex of the angle).
- The **Azimuth** is editable only for the starting element of the road because, by default, the azimuth is set tangent to the previous element.
- To change the azimuth of all other elements, clear the check mark from the **Tangent to Previous** box.
  Note*:* Use caution when setting the azimuth, since road elements are usually tangential to each other.

- Select the direction of **Turn**. The *Right* value stands for clockwise direction, the *Left* value for counterclockwise direction.
- The plot will show the element's appearance.

Spiral

The Spiral dialog allows you to add a spiral:

- Enter the **Radius** of the curve at the start and end points or one of the two parameters unambiguously defining the radius: *Chord Angle* or *Curve Angle*.
- Enter either the **Length** of the spiral element or *Spiral Constant*. The Spiral Constant is the square root of the product of the length and the radius of the spiral.
- The Azimuth is editable only for the starting element of the road because, by default, the azimuth is set tangent to the previous element.
- To change the azimuth of all other elements, clear the check mark from the ***Tangent to Previous*** box.
  Note*:* Use caution when setting the azimuth, since road elements are usually tangential to each other.
- Select the direction of **Turn**. The *Right* value stands for clockwise direction, the *Left* value for counterclockwise direction.
- Select the **Direction** of movement along the spiral, TS(traverse-spiral) to SC(spiral-circle) (entering the turn), or CS(circle-spiral) to ST(spiral-traverse) (exiting the turn), or CS to SC.
- The plot will show the element's appearance.

Intersection Point

The Intersect Pt dialog allows you to add an intersection point:

- Enter the name of the intersection **Point** or select it from the map or the list. Either enter the name manually or the North and East coordinates.
- **North** and **East** display the plane coordinates of the intersection point; they cannot be changed for an existing point.

- Enter the **Radius** of the curve or one of the two parameters unambiguously defining the radius: *Chord Angle* or *Curve Angle*.
- Enter either the **Length** or *Spiral Constant* for two spiral elements. The Spiral Constant is the square root of the product of the length and the radius of the spiral.
- The plot will show the element's appearance.

Edit Vertical Alignments

The dialog displays a list of the vertical alignments in the current job, and a preview window of the selected vertical alignment.

- Click **Add** to create a new vertical alignment. More...
- Click **Edit** to view/edit the currently selected vertical alignment. More...
- Click **Delete** to remove the selected vertical alignment from the job. You will be asked for confirmation.

Add Vertical Alignments

To create a new vertical alignment (VAL):

1. Enter a **Name** for the vertical alignment.
2. Select the **VAL Type**:
    - *Length & Grade:* select to create the vertical alignment by sections of graded lines. The vertical alignment is presented as a set of sections between the stations where the heights are known (usually these are the extremes of the vertical alignment line), and the interval around the station where the vertical alignment line has a parabolic shape.
    - *Station & Elevation:* select to create the vertical alignment by elements, starting and finishing at any station where you want and starting again.
3. Click to save settings and continue creating the vertical alignment.

Edit Vertical Alignments

In this dialog you can add/edit the elements of the Vertical Alignment and it's Start Point.

The icon brings up the pop-up menu of additional options. More...

Edit Vertical Alignment pop-up menu

If required, you can exchange roads:

- Select *Edit Points* to edit a point in the job. More...
- Select *High/Low Positions* to observe the stations of the highest and low positions in the vertical alignment. More...
- Select *Grade Display* to choose the display of grade values in the dialogs. More...

High/Low Positions

This dialog will display any extremums found along a vertical alignment. The types of extremums found are the high or low positions of arcs, curves and the transition points between two straight lines.

Alnt Name: The name of the vertical alignment.

The High/Low Positions list contains the following data:
Station: The station along the vertical alignment.
Type: The *High* or *Low* type of position.
Ell ht/Elevation (Unit): Ellipsodial height or Elevation, depending on the coordinate system used.
Radius (Unit): Radius of an arc with an high/low extremum.

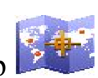
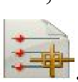
The icon  brings up the pop-up menu. Select the *Save to File* option to save the current list to a file with the default file name: "*HighLowPositions.txt*". You can edit the file name.

Grade Display

The Grade Display dialog allows you to choose the desired dimension for grade values: Percentage, Run:Rise, or Slope.

Start Point

The Start Pt tab allows you to edit the following parameters:

- In **Alnt Name**, the name of the vertical alignment.
- In **Start Pt**, the name of the start point that can be entered manually or chosen from the map  or from the list .
- The point code can be entered manually or chosen from the drop-down list for a new point. Click  to enter the point attributes.

- **Hight** displays the elevation of the point.
- In **Start Sta**, enter the starting station for the vertical alignment.

Vertical Alignment

The Vert tab shows the list of vertical alignment elements, the vertical alignment plot, and the start and end stations (or chainage) for each element.

- The element list data depends on the vertical alignment type:
  Element: the icon and the name of the element
  Length: the length of the element
  Start Grade, End Grade: the grades of the element, at the starting and ending points. For a Vertical grade element these values are the same.
  Station: the station distance
  Elevation: the elevation value on the station
  VC Length: the vertical curve length is the length of the interval near the station, where the alignment has a parabolic shape
- Click **Add** to select elements from the floating menu to add after the last element:
  Vertical grade, Parabola, or Circular arc for the Length & Grade type of the vertical alignment,
  Parabola, Circular arc, or Grade Point for the Station & Elevation type (more...).
- Click **Insert** to select elements from the floating menu to insert at the selected location in the list.
- Click **Edit** to change the properties of the selected element.
- Click **Delete** to remove the highlighted element from the vertical alignment.
- Use the arrow icon to hide/show the plot.

Add Vertical Grade

To add/edit a vertical grade to the vertical alignment of the Length & Grade type:

1. Enter the **Length** of the vertical grade element.
2. Enter the **Grade** of the element in percentage. If the grade is falling, the value should be set negative.
3. Observe the preview.

Add Parabola

To add/edit a parabola to the vertical alignment of the Length & Grade type:

1. When you edit an existing parabola element, you can change it for the Circular arc type in the **Curve Type** field.
2. Enter the **Length** of the parabola element.

3. Enter the **Start Grade** and **End Grade** of the element in percentage. If the grade is falling, the value should be set negative.
4. Observe the preview.

Add Circular arc

To add/edit a circular arc to the vertical alignment of the Length & Grade type:

1. When you edit an existing circular arc element, you can change it for the Parabola type in the **Curve Type** field.
2. Enter the **Arc Radius** of the circular arc element.
3. Enter the **Start Grade** and **End Grade** of the element in percentage. If the grade is falling, the value should be set negative.
4. Observe the preview.

Add Element

To add/edit an element to the vertical alignment of the Station & Elevation type:

1. Enter the end **Station** for the element.
2. Enter the **Elevation** at the station.
3. In **Curve Length** or **Arc/Grade Pt**, enter the length of the vertical parabola or circular arc, respectively. (It is assumed that the station is located in the middle of the interval.)
4. The **Previous/Next** field will show the calculated values of Grade and Length of the previous/next part of the vertical alignment.

Edit X-Section Templates

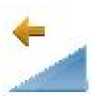
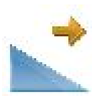
The X-Sect Templates dialog displays a list of the existing templates and a plot of the highlighted template.

The list contains three columns: Name (the name of the template), Cut Slope and Fill Slope values.

- Click **Add** to create a new template.
- Click **Edit** to change the properties of the selected template. More...
- Click **Delete** to remove the highlighted template from the list.
- Use the arrow icon to hide/show the plot.

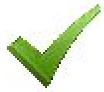
X-Sect Template

To add/edit an X-Sect Template:

- Enter the **Name** of the template.
- In the **Slope** field, enter the Cut and Fill values for the slope (Run values for cut and fill for a unit rise). These values represent the horizontal increment of the slope for a unit vertical increment. The Cut slope is used when the road surface is below the terrain, and the Fill Slope is used when the road surface is above the terrain.
- The dialog displays a list of segments comprising the template and a plot of the template. A list of segments consists of three columns:
  Segment Point: the name of the segment's end point on the x-section template
  Hz: the horizontal offset
  Vert: the vertical offset
- Use the icons  and  to show the template to right and left.
- Use the arrow icon to hide/show the plot.
- Use the buttons to edit the segments in the list:
  Add: to create a new segment that will be attached after the last segment in the list.
  Insert: to create a new segment that will be inserted in the list above the currently highlighted segment.
  Edit: to change the parameters of the highlighted segment. More...
  Delete: to remove the highlighted segment from the template.

Segment

To add/edit the template segment:

1. In **Segment Point**, enter the name of the segment end point.
2. In the **Offset** field:
   - Enter the *Horizontal* offset.
   - Press the *Down/Up/Grade/Ratio* button to select the type and enter the value of the vertical offset. Entered as Grade (in percentage) or as Ratio, the vertical offset will be recalculated to linear units after  is pressed.

Edit X-Sections

The X-Sect Set dialog displays a list of the existing x-section sets and a preview window of the highlighted x-section set.

- Click **Add** to create a new x-section set.
- Click **Edit** to change the properties of the selected x-section set. More...
- Click **Delete** to remove the highlighted x-section set from the list.

X-Section Set

The Edit X-Sect Set dialog contains a list of stations, where x-sections are applied. It also displays a preview of the highlighted x-section.

- The list of x-sections has the following columns:
  Station: The station where the x-section is applied.
  Left X Section, Right X Section: The names of the x-sections for the left and right parts of the road relative to the center line. The left and right cross sections can be different.
- Click **Add** to create a new x-section.
- Click **Edit** to change the properties of the selected x-section. More...
- Click **Delete** to remove the highlighted x-section from the list.

X-Section

To edit/add a cross section:

1. Enter the **Station** where the cross section will be applied.
2. From **X-Sect**, select which x-sections will be applied: *Left Only*, *Right Only*, or *Both*.
3. Select the **Left X-Section** and **Right X-Section** for the left and right parts of the road as required. These can be chosen only from the existing x-sections.
4. The preview window will show the edited x-section.

Edit String Set

The String Set dialog lists the names of the currently available String Sets. A preview window displays the currently selected String Set.

- Click **Add** to create a new string set.
- Click **Edit** to change the properties of the selected string set. More...
- Click **Delete** to remove the highlighted string set from the list.

Add/Edit String Set

To add/edit a new/existing String Set:

1. Enter the name of the **String Set**.
2. The **List of Strings** contains all Strings that comprise the String Set.
3. If required, use the up and down arrows to change the order of the Strings in the list.
4. The plot will display the currently selected String. It can be hidden with the arrow button.
5. Click **Delete** to delete all the selected Strings.
6. Click **Edit** to edit the currently selected String (double-clicking on the String will also edit it). More...
7. Click **Add** to add a new String to the current String Set.

Add/Edit Road String

To add/edit a new/existing Road String:

1. Enter the name of the **String**.
2. The list contains all pairs of the horizontal (Hz Alnt) and vertical (Vt Alnt) alignments that comprise the road string.
3. The plots will display the alignment pair currently selected. They can be hidden with the arrow button.
4. Click **Delete** to delete the selected pair from the Road String.
5. Click **Edit** to edit the alignment pair. More...
6. Click **Add** to add a new alignment pair to the current Road String.

Add/Edit Alignment Pair

This dialog allows you to create/edit a Road String's alignment pair in two ways:

1. From the **Hz Alnt** and **Vt Alnt** drop-down lists, select the horizontal and vertical alignments.
2. In the **Select** field, by creating the alignment pair from a linework:
    - Click *From List* to select a linework from the list. More...
    - Click *From Map* to select a linework from the map. More...

The icon  brings up the pop-up menu to edit:

- Horizontal alignments, select *Hz Alnt*. More...
- Vertical alignments, select *Vt Alnt*. More...

Select Linework from Map

Select a Linework on the map, which will be passed to the previous dialog for conversion or list selection. Note that you cannot select any lines other than Linework lines.

Calculate folder

Click an icon to calculate the task:

Point To Point
Computes the inverse (azimuth and distance) between two known points.

Point to Line
Computes the station of a known point inverse to a known line.

Point to Curve
Calculates the station of the known point inverse to the known curve.

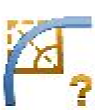
Point in Direction
Calculates the coordinates of a point, using a known point, and angle and distance offsets from the known point.

Intersection
Computes the intersection point or points given two known points, and either the direction or distance from the known point.

Calculator
Does calculations and conversion in scientific and standard modes. You type in the entire equation you want to calculate, press equals, and the result is shown.

Area by Points
Computes the area of a polygon with known vertices.

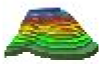
Corner Angle
Calculates the angle between two lines, a start line and an end line, which have a common mid point.



Curves
Computes a full set of parameters to determine a curve using four various minimum sets of specified parameters.

Offsets
Calculates the coordinates of points along a line, a curve, or a road.

Create Surface
Creates a new Surface.



[Contour](#)

Represents Surface data along contour lines.

Two-Point Inverse

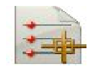
To calculate the inverse (azimuth and distance) between two known points:

1. In the **Input** tab, enter the known points manually or select them from the map or from the list of the job points.
   - In **From Point**, enter the first known point.
   - In **To Point**, enter the second known point.
2. Click the **Calc** button to calculate the inverse.
3. In the **Results** tab, observe the results of the calculation and click to save the data to a txt file if required:
   Azimuth: the azimuth from the first point to the second point.
   HDist: the horizontal distance from one point to another.
   VDist: the vertical distance from one point to another. The "-" sign means that the height of the second point is lower than the height of the first point.
   dNorth: the increment of the North coordinate.
   dEast: the increment of the East coordinate.
   dHeight: the increment of the height.
   Grade(Slope): the increment of the height in percent.
   Slope distance: the computed distance between the two points.
4. The **Map** tab shows the illustration for the results.

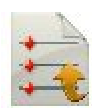
 Inverse Point to Line

To calculate a known point inverse to a known line:

1. In the **Input** tab:
   - Select the known **Point** name.
   - Select the **Start Point** of the line.
   - Select between **Azimuth** and **Az to Pt** to enter either the azimuth of the line manually or select another point to use the azimuth from the known point to this point as the line's direction.

     Note: Every point can be entered manually or selected from the map  or from the list  of the job points.
     When the second point is selected in *Az to Pt* to set the line, you can select the **Store PTL Point** check box to save PTL data of the known point if required.
   - Set the **Start Station** of the line.
   - The **COGO Pt** field will display the default name of the known point's projection on the line. You can change this name.

     Select the code for this point from the existing codes and, if required, click  to set the point's attributes . [More...]
   - Click the **Calc** button  to calculate the inverse.

2. In the **Results** tab, observe the results of the calculation and click  to save the data to a txt file if required:
   Projected Point: the name of the known point's projection on the line.
   North, East, Elev: the coordinates of the projected point.
   From Point: the name of the known point.
   The *Line* information:
   Start Point: the name of the starting point of the line.
   Tangent Azimuth: the azimuth of the known line.
   Projected Azimuth: the azimuth of the perpendicular from the known point to the line.
   Start Station: the starting station of the line.
   Station: indicates the distance between the starting point and the projection of the known point on the line.
   Offset: the horizontal offset between the known point and the projected point.
   Height: the vertical offset between the known point and the projected point.
3. The **Map** tab shows the illustration for the results.

Inverse Point to Curve

To calculate a known point inverse to a known curve:

1. In the **Input** tab, select points needed for calculations. Every point can be entered manually or selected from the map or from the list of the job points:
   - Select the known **Point** name.
   - Switch between **PC** and **RP** to enter the first curve point as required. Depending upon the first curve point chosen, you can define the curve by two different sets of points:
   - Enter either **PC** point, **Curve** point, **PT** point.
   - Or **RP** point, **PC** point, **PT** point.
     In this case, the distance between **RP** point and **PC** point should be equal to the distance between **RP** point and **PT** point, and two curves can be created: a *Small* curve of 180 degrees or less and a *Large* curve of 180 degrees or more.
   - From the **Curve** drop-down list, select which of these two curves should be used for computations.
   - The **COGO Pt** field will display the default name of the known point's projection on the curve. You can change this name.

     Select the code for this point from the existing codes and, if required, click to set the point's attributes . More...
   - Set the **Start Station** of the reference curve.
   - Click the **Calc** button to calculate the inverse.
2. In the **Results** tab, observe the results of the calculation and click to save the data to a txt file if required:
   Projected Point: the name of the known point's projection on the curve.
   North, East, Elev: the coordinates of the projected point.
   Point: the name of the known point.
   The *Curve* information: *PC Point*, *Curve Point*, *PT Point.*
   Tangent Azimuth is the azimuth of the tangent of the curve at the point of the known point's projection.
   Projected Azimuth: the azimuth of the perpendicular from the known point to the tangent of the curve.
   Start Station: the starting station of the curve.
   Station indicates the distance between the starting station and the known point's projection on the curve.
   Offset is the horizontal offset between the known point and the projection point.

dHeight is the vertical offset between the known point and the projection point.

3. The **Map** tab shows the illustration for the results.

Calculate Curves

Click an icon to calculate the task:

Curve
Calculates the full set of curve parameters, given one length and one radius parameter.

3Pt Curve
Computes the curve parameters given three points: starting point of the curve (PC point), any curve point and ending point of the curve (PT point), the Radius point, and PC and PT points.

PI & Tangents
Computes the starting, ending and center points of a curve, given the point of intersection, the radius, and the azimuths from the PI point to the PC and PT points respectively.

Radius & Points
Computes the parameters and the coordinates of the center of a curve given the starting and ending points of the curve and a radius parameter.

Curve Solution

To calculate the full set of curve parameters when a curvature parameter and a length parameter are given:

1. In the **Input** tab:
   - Select one of the curvature parameters **Radius/Chord Angle/Curve Angle** and enter the value of it.
   - Select one of the length parameter of the curve **Delta/Length/Chord/Tangent/Mid Ord/External** and enter the value of it.
   - Observe the plot of the curve in the view window.
   - Select the *Right/Left* direction of **Turn** relative to the starting point.
   - Click the **Calc** button to calculate the parameters of the curve.
2. In the **Results** tab, observe the results of the calculation and click to save the data to a txt file if required:

   Radius of the curve.
   Length of the curve.
   Chord is the distance between the starting and ending points (tangent points) of the curve.
   Degree Curve defines the angle in degrees which is used to compute the radius of a curve with a length of 100 units.
   Degree Chord defines the angle in degrees which is used to compute the radius of curve whose chord is 100 units long.
   Delta is the internal angle from center to tangent points.
   Tangent is the distance between the point of intersection of the tangents at the starting and ending points, and the ending point.
   External is the shortest distance between the point of intersection and the curve (along the line joining the point of intersection to the radius point).
   Mid Ord is the length of the line segment between the curve and the chord on the line joining the point of intersection to the radius point.
   Segment is the area of a circle bounded by a chord and the minor arc that it cuts off.
   Sector is the area of a circle bounded by two radii and the minor arc they determine.
   Fillet is the area between the arc of a circle and the two tangents at the end points of the arc.
3. The **Map** tab shows the illustration for the results.

Three-Point Curve

To calculate the curve parameters when three points are given:

1. In the **Input** tab, select points needed for calculations. Every point can be entered manually or selected from the map or from the list of the job points:
   - Switch between **PC Point** (point of curvature) and **RP Point** (radius point) to enter the first curve point as required. Depending upon the first curve point chosen, you can define the curve by two different sets of points:
   - Enter **PC** point, **Curve** point, **PT** point (point of tangency).
     In this case, the coordinates of the **RP Point** will be calculated along with the curve parameters. The default name will be shown that can be changed. Select the code for this calculated point from the existing codes and, if required, click to set the point's attributes . More....
   - Or **RP** point, **PC** point, **PT** point.
     In this case, the distance between **RP** point and **PC** point should be equal to the distance between **RP** point and **PT** point, and two curves can be created: a *Small* curve of 180 degrees or less and a *Large* curve of 180 degrees or more. From the **Curve** drop-down list, select which of these two curves should be used for computations.
   - Click the **Calc** button to calculate the curve parameters.
2. In the **Results** tab, observe the results of the calculation and click to save the data to a txt file if required:
   North, East, Elev: the coordinates of the RP point.
   Radius of the curve.
   Length of the curve.
   Chord is the distance between the starting and ending points (tangent points) of the curve.
   Curve Angle defines the angle in degrees which is used to compute the radius of the curve with a length of 100 units.
   Chord Angle defines the angle in degrees which is used to compute the radius of the curve whose chord is 100 units long.
   Delta is the internal angle from center to tangent points.
   Tangent is the distance between the point of intersection of the tangents at the starting and ending points, and the ending point.
   External is the shortest distance between the point of intersection and the curve (along the line joining the

point of intersection to the radius point).

Mid Ord is the length of the line segment between the curve and the chord on the line joining the point of intersection to the radius point.

Segment is the area of a circle bounded by a chord and the minor arc that it cuts off.

Sector is the area of a circle bounded by two radii and the minor arc they determine.

Fillet is the area between the arc of a circle and the two tangents at the end points of the arc.

3. The **Map** tab shows the illustration for the results.

PI & Tangents

To calculate the curve parameters when two tangents and the point of their intersection are given:

1. In the **Input** tab, enter the data needed for the task:
    - Select the **PI Point** (Point of Intersection). It can be entered manually or selected from the map or from the list of the job points.
    - In **Az PI to PC**, enter the azimuth from the PI point to the starting curve point.
    - In **Az PI to PT**, enter the azimuth from the PI point to the ending curve point.
    - In **Radius/Curve Angle/Chord Angle/Tangent**, enter the appropriate radius parameter of the curve.
    - Enter the **PC Point** name and select the code for the calculated starting curve point.
    - Enter the **PT Point** name and select the code for the calculated ending curve point.
    - Enter the **RP Point** name and select the code for the calculated radius point.
    - Click to set the point's attributes if required. More...
    - Click the **Calc** button to calculate the curve parameters.
2. In the **Results** tab, observe the results of the calculation and click to save the data to a txt file if required:

    North is the northing coordinate of the PC/PT/RP points.
    East is the easting coordinate of the PC/PT/RP points.
    Elev is the height of the PC/PT/RP points.
    Radius of the curve.
    Length of the curve.
    Chord is the distance between the starting and ending points (tangent points) of the curve.
    Curve Angle defines the angle in degrees which is used to compute the radius of a curve with a length of 100 units.
    Chord Angle defines the angle in degrees which is used to compute the radius of the curve whose chord is 100 units long.
    Delta is the internal angle from center to tangent points.
    Tangent is the distance between the point of intersection of the tangents at the starting and ending points, and the ending point.
    External is the shortest distance between the point of intersection and the curve (along the line joining the point of intersection to the radius point).
    Mid Ord is the length of the line segment between the curve and the chord on the line joining the point of

intersection to the radius point.

Segment is the area of a circle bounded by a chord and the minor arc that it cuts off.

Sector is the area of a circle bounded by two radii and the minor arc they determine.

Fillet is the area between the arc of a circle and the two tangents at the end points of the arc.

3. The **Map** tab shows the illustration for the results.

Radius & Points

To calculate the parameters of a curve when the starting/ending points and the radius parameter of the curve are given:

1. In the **Input** tab, enter the initial data for the task. The points can be entered manually or selected from the map or from the list of the job points.
    - Set the **PC Point** (point of curvature).
    - Set the **PT Point** (point of tangency).
    - In **Radius/Curve Angle/Chord Angle**, enter the appropriate radius parameter of the curve.
    - Select the *Left* or *Right* direction of **Turn**, relative to the PC Point.
    - Select the **Curve** in the circle that should be considered. The radius, and the PC and PT points define two curves: one with delta less than or equal to 180 degrees (*Small* curve), and the other with delta greater than or equal to 180 degrees (*Large* curve).
    - The coordinates of the **RP Point** will be calculated along with the curve parameters. The default name will be shown that can be changed. Select the code for this calculated point from the existing codes and, if required, click to set the point's attributes. More....
    - Click the **Calc** button to calculate the curve parameters.

2. In the **Results** tab, observe the results of the calculation and click to save the data to a txt file if required:
   North is the northing coordinate of the RP point.
   East is the easting coordinate of the RP point.
   Elev is the height of the RP point.
   Radius of the curve.
   Length of the curve.
   Chord is the distance between the starting and ending points (tangent points) of the curve.
   Curve Angle defines the angle in degrees which is used to compute the radius of the curve with a length of 100 units.
   Chord Angle defines the angle in degrees which is used to compute the radius of the curve whose chord is 100 units long.
   Delta is the internal angle from center to tangent points.
   Tangent is the distance between the point of intersection of the tangents at the starting and ending points, and the ending point.
   External is the shortest distance between the point of intersection and the curve (along the line joining the

point of intersection to the radius point).

Mid Ord is the length of the line segment between the curve and the chord on the line joining the point of intersection to the radius point.

Segment is the area of a circle bounded by a chord and the minor arc that it cuts off.

Sector is the area of a circle bounded by two radii and the minor arc they determine.

Fillet is the area between the arc of a circle and the two tangents at the end points of the arc.

3. The **Map** tab shows the illustration for the results.

Calculate Offsets

Click an icon to calculate the task:

Line Offset
Defines a line between two points used for computing point locations relative to the line.

Corners Offset
Defines a point location relative to the nodes of the polyline.

Curve Offset
Defines a curve (section of an arc) used for computing point locations in relation to the curve.

Polyline Offset
Defines a linework used for computing point locations in relation to the polyline.

Road Offset
Calculates point locations relative to the selected road or alignment.

Line Offset

The line is defined by its azimuth, azimuth to another point, or the end point of the line. The plot will display the defined line.

To calculate point locations relative to a line:

1. Enter the **Start Point** of the line. This can be entered manually, or selected from the map or from the list of the job points.
2. Select between **End Point** and **Azimuth** to define the direction of the line.
3. In **Ht Comp**, select the type of height computations for the computed points:
    - *Ht of Start Pt (height of the starting point):* The computed points will have the same height as the starting point of the line.
    - *Interpolate Ht:* The height of the computed points will be computed through linear interpolation using the height of the starting and ending points of the line.
4. Select the **Num Subs** check box to enter the number of subdivisions if required to subdivide the line. For instance a value of 3 indicates that you want to compute four points by subdividing the line in three equal segments.
5. Select the **Include Transition Point** check box to include transition points if they do not fall on a designated station.
6. In **SS**, enter the starting station (chainage) of the line.
7. Click **Next** to to define stationing for offsets and calculate points. More...

Station & Offsets

To define stationing which is used to compute the points locations in relation to lines, curves and alignments:

1. Select between **Station** and **Start Station** to set the current or starting station. The two arrow buttons allow you to decrease or increase the station by the interval specified in the station **Interval** shown.
2. Set the desired **End Station** if you wish to compute several points. The two arrow buttons allow you to decrease or increase the station by the interval specified in the Station Interval shown in the next line.
3. Set the station **Interval**.
4. Enter the **Right Offset** or **Left Offset** of the computed point with respect to the line at the stations.
5. Enter the **Up** or **Down** height offset, or the vertical **Grade** (in percentage) with respect to the height of the line at the station. If the grade is falling, the value is set negative.

6. Enter the name of the current or starting computed **COGO Point**. Select the code for this point from the existing codes and, if required, click to set the point's attributes . More...
7. Click **Calc** to calculate the offset points.

Corners Offset

New points are defined by the offset from the nodes of the existing polyline. Offset may be defined in three ways – either inline, or perpendicular right or perpendicular left. Note, that the new point will be created for each polyline segment, connected to the node.

To calculate point locations in relation to a polyline nodes:

1. Choose between two options to define the polyline:
   - **Polyline**: Select an existing polyline from the polylines list or select the polyline in the map . The plot window will display the defined polyline.
   - **Code**: Define a code generated line. When Code is selected, the drop-down list will contain a list of all polyline codes used in the job which have associated strings. The list will display all of the available strings associated with the selected code. If the user selects a string the plot window will update to show the selection.
2. Click **Next**.
3. Select between **Perpendicular/Inline** and enter the offset from the linework node.
4. If selected **Perpendicular**, select either **Left** or **Right**.
5. In **COGO Pt**, enter the name of the first point to be created.
6. In **Code**, select the code for the points to be created.
7. Click **Create**.
8. The software creates points with the defined offset.

Curve Offset

To calculate point locations in relation to a curve:

1. In **PC Point**, enter the point of curve, the starting point of the arc.
2. In **PT Point**, enter the point of tangency, the ending point of the arc.
3. Select between **Radius/ Curve Angle/ Chord Angle** and enter the radius parameters of the curve as required.
4. In **SS**, enter the starting station (chainage) of the curve.
5. In **Ht Comp**, select the type of height computations for the computed points:
   - *Ht of Start Pt (height of the starting point):* The computed points will have the same height as the starting point of the line.
   - *Interpolate Ht:* The height of the computed points will be computed through linear interpolation using the height of the starting and ending points of the line.
6. In **Turn**, select on which side the center point of the curve is located. The plot will display the defined curve, and **Length** will show the calculated length of the curve.
7. Select the **Include Transition Point** check box to include transition points if they do not fall on a designated station.
8. Length, enter the starting station (chainage) of the line.
9. Select the **Num Subs** check box to enter the number of subdivisions if required to subdivide the line. For instance a value of 3 indicates that you want to compute four points by subdividing the line in three equal segments.
10. Click **Next** to define stationing for offsets and calculate points. More...

Polyline Offset

Polyline is provided in the points which are connected to form opened or closed polylines.

To calculate point locations in relation to a polyline:

1. Choose between two options to define the polyline:
   - **Polyline***:* Select an existing polyline from the polylines list or select the linework in the map . The plot window will display the defined polyline.
   - **Code***:* Define a code generated line. When Code is selected, the drop-down list will contain a list of all polyline codes used in the job which have associated strings. The list will display all of the available strings associated with the selected code. If the user selects a string the plot window will update to show the selection.
2. Select a task you want to calculate:
   - **Offset Polyline***:* To create an offset polyline with respect to the selected linework, click **Next**. More...
   - **Offset Points From Polyline***:* To create offset points from the selected line, select the start station and the interval and click **Next**. More...
3. Click **Create Points** if required to generate points along a polyline and save them in a point list. More...

Offset Polyline

To create a new **Offset Polyline** at entered offsets from the selected linework:

1. In the **Offset Line** field enter the name for the offset linework. The field initially displays a default name <*the linework name_[n]*> where *[n]* is an integer for the first available unique name that does not already exist in the job. You can change this name.
2. Enter the **Right** or **Left Offset** of the computed line with respect to the linework.
3. Enter the distance **Up** or **Down** the new line will be shifted. Distance units associated with the edit fields are the current job distance units displayed.
4. Observe the preliminary view in the horizontal and vertical planes. The plot windows show both the selected line as well as the offset line at the current entered offset values. These plot windows update dynamically every time you change the offset values.
5. Click **Create** to compute and save the offset linework to the current job.

Offset Points From Polyline

To create points at the entered offsets from the selected linework:

1. In **SS**, enter the starting station (chainage) of the linework.
2. Select the **Include Transition Point** check box to create transition points every time the stationing passes a transition point.
3. In **Interval**, select how the offset points will be created.
   - **Interval**: Enter the station interval to subdivide the linework by entering an interval value in the next dialog. More...
   - **Subdivide**: Enter a number of subdivisions by which the linework will be split.
   - **Transition Only**: The offset points will only be created at transition points between the start and end stations.
4. Click **Next** to define stationing for offsets and calculate points. More...

Station & Offsets

To define stationing which is used to compute the points locations in relation to lineworks:

1. Select between **Station** and **Start Station** to set the current or starting station. The two arrow buttons allow you to decrease or increase the station by the interval specified in the station *Interval* shown. If tou had selected *Include transition points* (More...) incrementing and decrementing the stationing will also include the transition point stations. If you selected the *Subdivision* or *Transition Only* the Station button will be hidden to enter the start and end station where the offset points will be created.
2. Set the desired **End Station** if you wish to compute several points. The two arrow buttons allow you to decrease or increase the station by the interval specified in the station *Interval* shown in the next line.
3. Enter the station **Interval**, the interval at which the stationing will increase or decrease.
4. Enter the **Right Offset** or **Left Offset** of the computed point with respect to the line.
5. Enter the **Up** or **Down** height offset, or the vertical **Grade** (in percentage) with respect to the height of the linework. If the grade is falling, the value is set negative.
6. In **Angle**, select how the angle points will be computed at segment intersections:
   - **Offset Bk**: The offset point at the intersection will be computed from the end point of the first segment.
   - **Offset Ahd**: The offset point at the intersection will be computed from the start point of the second segment.
   - **Bisector**: The offset point at the intersection will be computed from the intersection of the offset of the first and second segment.
7. In **Curve**, select how points on curve segments will be selected:
   - **Interval**: The offset points on curve segments will be computed at intervals along the curve.
   - **RP**: Only the curves radius point will be computed.
   - **PI**: Only the curves point of intersection of tangents will be computed.
   - **MOC**: Only the point in the middle of the curve will be computed.
8. Enter the point information for the first created **COGO Point**. Select the code for this point from the existing codes and, if required, click

 to set the point's attributes . More...

All subsequent created points will be increments from the entered point name and have the same code values.

9. Click **Calc** to calculate and save the offset point locations to the job.

   If the you have hidden the end point stationing, only a single point will be created at the current stationing. If you have entered the end point stationing, points will be created between the start and end station at the current entered interval including transition points if you have selected to include them. If you have selected to subdivide the line, points will be created from the start station to the end of the line.

Create Points

To create points along a linework:

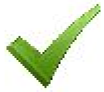
1. Select the **Interval** or **Segments** radio button to enter the interval between the calculated points or the number of segments in the linework.
2. Select the **Along Tangent** check box to enter the interval between the calculated points or the number of segments in a straight line.
3. Select the **Along Curve** check box to enter the interval between the calculated points or the number of segments in a curve.
4. Select the corresponding check boxes to create additional linework points as required:
   - **End Points**: The linework's start and end points.
   - **Curve PIs**: The points of intersection of tangents to the curves drawn at the start and end curve points.
   - **Curve RPs**: The radius points.
   - **Curve MOCs**: The points in the middle of the curves.
5. Click **Next** to define the calculated point's details. More...

Points Details

Enter details for the calculated points:

1. Enter the name of the **First Point**. Select the code for this point from the existing codes and, if required, click  to set the point's attributes . More...
2. In required, select the **Prefix/Suffix** from the drop-down list and enter the desired value.
3. Optionally, select the **Save Points To Point List** check box to create a Point List with the name specified in

the field which appears.

4. Click  to save the calculated points.

Road Offset

To caculate the offset points along an alignment:

1. In **Road/H Alnt/HV Alnt**, enter the name of the road or alignment to be used for computing points. This can be entered manually, or chosen from the list .
2. The **SS** field shows the starting station of the selected road of alignment, the distance from the beginning. The plots display the alignment in the horizontal and vertical planes.
3. If required, click **Calculate Road Points** to generate points along the defined Road. More...
4. Click **Station & Offsets** >> to define stationing for offsets and calculate points. More...

Compute Area By Points

To calculate the area of a polygon with known vertices:

1. In the **Input** tab:
   - Select the name of the **PointList**, **Linework** or **Area** that contains vertices of the polygon. The name can be entered manually or selected from the list .

     The points of the selected Point List will be listed along with their codes in the table and the polygon will be shown in the plot. Use the arrow button to hide/show the plot as required.
   - The Up and Down arrow buttons can be used to modify the order of the points to obtain the correct shape of the polygon.
   - Click the **Calc** button to calculate the area of the polygon.
2. In the **Results** tab, observe the results of the calculation and click to save the data to a txt file if required:
   The calculated Area in (JobUnits)^2, acres and hectares.
   Perimeter of the polygon.
   The tolerance interval calculated as Area (in meters) minus/plus Perimeter multiplied by 1.25.
   The list of points constituting the polygon vertices in the correct order.
3. The **Map** tab shows the illustration for the results.

Surface Volume

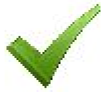
To perform volume calculations:

1. In the **Final** field, select the Surface on which you want the volume calculations to be performed. Either manually enter the name of an existing Surface in the edit field or press the List selection button and choose the Surface from the list. When the Surface selection changes an information message will display the maximum and minimum Northing and Easting values for the area covered by the Surface.
2. From the **Boundary** drop-down menu, select one of six options for volumes calculations.
   - **Min Elevation:** the plane of projection of the final Surface will be defined as a flat plane at a minimum elevation.
   - **Max Elevation:** the plane of projection of the final Surface will be defined as a flat plane at a maximum elevation.
   - **Boundary:** the plot will display only the final Surface. The plane of projection of the final Surface will be defined as a flat plane at a fixed elevation of zero.
   - **Fixed Elevation:** the plot will display only the final Surface. The plane of projection of the final Surface will be defined as a flat plane at a fixed elevation. Either manually enter the fixed elevation in the edit field or press the Map or the List selection button to select a point in the job whose elevation will be used as the fixed elevation. After the point selection dialog has closed the edit field will display the elevation of the selected point.
   - **Plane:** the plane of projection of the final Surface will be defined by three separate points. Either manually enter the names of the three points in the edit fields or use the associated Map and List selection buttons to select these points from the job.
   - **Original Surface:** select another existing Surface. Either manually enter the name of the second Surface or press the List selection button and choose the Surface from the list. The plots will display both of the Surface selected.
3. Press the **Calc** button to calculate the cut and fill volumes and projected areas of a Surface to another Surface or a plane of projection.

When a successful Surface calculation has been performed the Results tab will display the calculation data. A warning message displays if any of the fields is empty or contains a name that does not exist.

Open Surface

To open an existing Surface:

1. Highlight the name of the TIN file you want to open.
2. Click  to open the file.

Create Surface

Create Surface allows you to create a Topcon *tn3* Tin by selecting points from the job, existing point lists, existing areas, existing linework. Surface is a three-dimensional display of transformed elevation data. Surface describes the topographic surface with a network of digitized points and optional breaklines.

To create a Surface

1. In the **Points/Point List/Linework/Area** field, select the current method of selection of data required to create the Surface.
   - For **Points,** in the **Sel Pts** drop-down list, choose an option to select multiple points in the job for creating the Surface:

   - *All* - all the job points.
   - *By Range* - points from a range. More...
   - *By Code* - points of a selected code. More...
   - *By Code String* - points of a selected code string. More...
   - *By Radius* - points around a selected point at a certain distance. More...
   - *From Map* - points selected graphically from the map More...
   - *From List* - points from a list of points. More...
   - *From Layer* - points on a selected layer. More...

   - For **Point List**, press the List selection button and select an existing point list in the job. More... The list name will appear in the edit field. The name can also be manually entered into the edit field and added to the point selection by pressing the ok button.
   - For **Linework** and **Area**, press either the Map selection button and select an existing linework/area from the map (more...) or press the List selection button and select an existing linework/area from the list (more...). The name will appear in the edit field. The name can also be manually entered into the edit field and added to the point selection by pressing the ok button.
2. The list will display the points currently selected, and the plot will display the created Surface.
3. To obtain information about a single point highlighted in the list, use the Info button.
4. If required, check mark the **Include Breaklines** box to apply breaklines and exclusion areas to the Surface. Breaklines are linear elements that describe changes in smoothness or continuity of the surface.

   Note: The points whose feature code is flagged as being a breakline or exclusion area get applied to the Surface in the order that they appear in the point list. You can alter the exclusion areas and breaklines by moving their points up or down in the list.

5. Click **Save** to save the Surface created from the selected points in the Surface folder by default. The default name will be *newTIN.TN3*.

6. When a valid Surface has been created, the **Results** tab displays the results updated dynamically as you alter the data.
7. The Map tab displays the created Surface as the current selected points and the triangulation lines.
8. A warning message displays if less than three or no points have been selected, or there are not enough points to create a Surface since several have been excluded because their code was flagged as not to be used in a Surface or they are incomputable in the current system.

To edit the Surface

1. To view a point on the plot, highlight the point in the list.
2. To see a Map view of the current Surface, double-click in the plot window.
3. To delete any point from the Surface:
   - Highlight the point in the list.
   - Click the Delete button. The Surface will be updated.
4. To alter the boundary of the currently selected Surface:
   - Press the **Edit Boundary** button. Find out more...

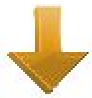
Icons you can use

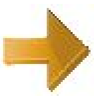
MAGNET Field Site general buttons and icons are described here...

General Icons

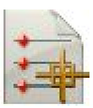
 The Delete button is used to delete the selected points in the point list.

 The Info button is used to display information on a single point highlighted in the list.

 The Up (and Down ) arrow button is used to move the selected points up (and down) in the list control. If the current selection is at the top or bottom or nothing is selected the buttons will be disabled accordingly.

 The Plot button is used to hide and show the plot window.

 The Arrow control button allows you to use the keyboard arrow keys to move points inside the list.

 Opens the list of points.

 Opens a map.

 Confirms settings, closes the dialog, and returns to the previous dialog.

Editing Boundary

The boundary selection dialog contains a list of points, which bound the Surface, the general plot of the Surface with the boundary in red.

To edit the boundary:

1. If required, move points up and down the list to change the order of the boundary. The Surface will be updated to reflect the changes to the point order.

2. To delete a point from the boundary, highlight the point in the list and click the red minus icon. The Surface will be updated.
3. Use the Map or List selection button to select points from all of the points in the Surface to add to the boundary.

   Note: When you select a point in the boundary Map selection dialog the point is added between the end points of the segment which is closest to the selected point. After each selection the Surface gets updated to show your selection. When you return to the Edit Boundary dialog the point list gets updated to the order of the boundary in the map.

Find out description of the icons on the dialog here...

Contour Surface

This dialog allows you to contour an existing Topcon *TN3 TIN*. Contouring Surface is a method of transforming elevation data into a contoured surface, representing data along contour lines.

To create the contours on the Surface:

1. Enter the *Surface Name* manually in the edit field or press the List selection button to select an existing Surface from the list.
2. The plot window will display the currently selected Surface.
3. Press the **Next** button to set the parameters required for contouring the Surface.
4. Enter an *Contour Interval* between the contour lines in the edit field as required. The Contour line interval must be greater than 0.001 m. The difference between the start and end elevations is divided by the interval to get the number of contour lines.
5. The *Start Elevation* edit field initially shows the lowest elevation in the selected Surface. If required, you can enter a different value.
6. The *End Elevation* edit field initially shows the highest elevation in the selected Surface. If required, you can enter a different value.
7. Press the *Save* button to store the contoured Surface and generate the contour lines in the job. The lines will be added to the job as lines with the Surface name and an integer value starting at 1 which makes the line unique.

The results of contouring will be displayed on the *Results* tab.

The *Map* tab will display the current selected Surface as well as the contour lines in red.

Point In Direction

To calculate the coordinates of a point in a certain direction from a known point:

1. In the **Input** tab:
    - In **From Point**, enter the known point manually or select it from the map or from the list of the job points.
    - Select between **Azimuth** and **Az to Pt** to enter the azimuth at the known point either by value or as a direction to another known point. To automatically add/subtract 90 or 180 degrees, click and select the desired action.
    - Enter the offsets from the known point:
        - Set the **Angle Offset** from the azimuth line.
        - In **HD Gnd/HD Grid** (depends on the current coordinate system), set the horizontal distance offset along the angle offset line.
        - In **Vert Dist**, set the height offset.
    - The **COGO Pt** field will display the default name of the unknown point. You can change this name. Select the code for this point from the drop-down list. Click to set the point's attributes . More...
    - Click the **Calc** button to calculate the point.
2. In the **Results** tab, observe the results of the calculation and click to save the data to a txt file if required:
   The coordinates of the unknown point:
   North is the northing coordinate of the point.
   East is the easting coordinate of the point.
   Elev is the height of the point.
   From Point is the name of the known point.
   Azimuth from the known point to the unknown point.
3. The **Map** Tab shows the illustration for the results.

Intersection

To calculate the intersection point(s) of two given directions or two distances:

1. In the **Input** tab:
   - Select the name of the known **Point 1**. The point can be entered manually or selected from the map or from the list of the job points.
   - Select between **Azimuth** and **Az to Pt** to enter a direction from the Point 1 to the unknown intersection point. Either enter the azimuth to the unknown point manually or select another point to calculate the azimuth from the Point 1 to this point and use it as the direction.
     Select **Distance** to enter a distance to the unknown point. In this case, there will be two solutions of calculation and two intersection points will be determined.
   - Select the name of the known **Point 2**.
   - Select between **Azimuth** and **Az to Pt** to enter the direction from the Point 2 to the unknown point or select **Distance** to enter a distance to the unknown points.
   - The **COGO Pt** field will display the default name of the intersection point. You can change this name.

     Select the code for this point from the drop-down list. Click to set the point's attributes . More...
   - Click the **Calc** button to calculate the intersection point.
2. In the **Results** tab, observe the results of the calculation and click to save the data to a txt file if required:
   North, East, Elev: the coordinates of the first/second found point.
3. The **Map** tab shows the illustration for the results.

Calculator

The calculator is a powerful tool to do all sorts of calculations and conversions. Type in the entire equation you want to calculate, press equals, and the result is shown:

- *Input Field* is where calculations are done. It will accept an entire calculation and then once equals is pressed it will calculate everything at once.
- *Result Field* is where the result is shown once equals is pressed. This field is also used as the y or theta values for rectangular/polar conversions.
- *Previous Result Field* is where the previous result is moved up to once equals is pressed. This field is also used as the x or r values for rectangular/polar conversions.

The calculator operates in two modes:

- Click the **std** button to open the standard panel. Standard mode performs common math functions.
- Click the **sci** button to open the scientific panel. Scientific mode performs more complex scientific functions.

Standard Calculator

The calculator in standard mode allows you to perform standard maths.

Learn descriptions of the functions:

square root	Calculates the square root of a value.
x squared	Calculates the square of a value.
inverse	Calculates the inverse of a value or 1/x.
nth power	Calculates the nth power of a value in the format x ^ power.
percent	Converts the value to a percent or x/100.
MC	Clears the memory.
MR	Recalls the memory value indicated by M in the input field.
MS	Saves the already computed result into memory.
M+	Adds the already computed result to the value in memory.
C	Clears out all the fields.
<-	Backspace, removes the last entry.

copy	If the calculator was started from an edit field, it copies the value back to that field.
sci	Brings up the scientific calculator.
+/-	Alternates the value between positive and negative.

Scientific Calculator

The calculator in scientific mode allows you to perform scientific functions.

Learn descriptions of the functions:

sin	Calculates the sine.
cos	Calculates the cosine.
tan	Calculates the tangent.
arcsin	Calculates the arcsine.
arccos	Calculates the arccosine.
arctan	Calculates the arctangent.
log	Calculates the logarithmic value base 10.
ln	Calculates the natural logarithmic value or base e.
square root	Calculates the square root of a value.
x squared	Calculates the square of a value.
inverse	Calculates the inverse of a value or 1/x.
nth power	Calculates the nth power of a value in the format x ^ power.
percent	Converts the value to a percent or x/100.
MC	Clears the memory.
MR	Recalls the memory value indicated by M in the input field.

MS	Saves the already computed result into memory.
M+	Adds the already computed result to the value in memory.
Rad Deg	Converts radians to degrees.
Deg Rad	Converts degrees to radians.
Grad Deg	Converts gradians to degrees.
Deg Grad	Converts degrees to gradians.
Deg DMS	Converts degrees to degrees minutes seconds notation.
DMS Deg	Converts degrees minutes seconds notation to degrees.
Rec Pol	Converts rectangular coordinates (xy) to polar (r theta). This is done by using both result fields, the top one being x, bottom one being y. Enter the values and click Rec Pol and it will convert the top value to r and bottom to theta.
Pol Rec	Converts polar coordinates (r theta) to rectangular (xy). This is done by using both result fields, the top one being r, bottom one being theta. Enter the values and click Pol Rec and it will convert the top value to x and bottom to y.
DMS+	Adds the left and right values assuming both are in DMS form.
DMS-	Subtracts the right value from the left assuming both are in DMS form.
e	The constant e, base of the natural logarithm.
pi	The constant pi, the circumference to diameter ratio for any circle.
C	Clears out all the fields.
<-	Backspace, removes the last entry.
copy	If the calculator was started from an edit field, it copies the value back to that field.
std	Brings up the standard calculator.
Deg	If this is showing, it means the calculator is working in degrees.

Grad If this is showing, it means the calculator is working in gradians.

Rad If this is showing, it means the calculator is working in radians.

+/- Alternates the value between positive and negative.

Corner Angle

To calculate the angle between two lines, which have a common mid point:

1. In the **Input** tab:
   - Select the **Start Point** that defines the first side of the angle.
   - Select the **Mid Point** that defines the corner of the angle.
   - Select the **End Point** that defines the second side of the angle.

     Note: Every point can be entered manually or selected from the map or from the list of the job points.
   - Click the **Calc** button to calculate the angle.
2. In the **Results** tab, observe the results of the calculation and click to save the data to a txt file if required:
3. The **Map** tab shows the illustration for the results.

Map

The Map screen displays the map of the current job. To move the map display, hold down and slide the stylus on the screen. The Map maintains the scale after changing the status of the main map.

The basic commands of the Map screen are available through:

- The map view tools which are arranged in two groups. More...
- The pop-up menus which depend on the objects selected. More...
- The drawing toolbar which allows you to create an object. More...
- The snap toolbar which allows you to create a point for the object which was selected in the Drawing Toolbar. More...

Map View Tools

The tool bar may be opened/hidden by using the arrow icon.

Click a tool icon to execute the command:

Zoom In
Zooms the plot inwards.

Zoom Out
Zooms the plot outwards.

Zoom Window
Selects an area to center on. You may draw an area from low-right to top-left to highlight the object(s) you want.

Zoom All
Displays all objects in the map.

Zoom By Point
Selects a point for centering the plot.

Properties
Selects the map properties for display. The Map Properties are also available from every pop-up menu.

Map Pop-Up Menus

The pop-up menus in the Map screen contains commands which virtually duplicate the number of commands found in MAGNET Field Site and allow you to edit job data, to calculate an appropriate COGO task and to stake desired objects

To open a pop-up menu tap the object to highlight it. Hold down the stylus on the selected object until a pop-up menu displays. The menu options depend on the object selected. Use an item from the pop-up menu as required.

If you hold down on multiple objects, you obtain the Objects near selected point dialog that allows you to select individual objects.

Map Properties

The Map Properties dialog contains four tabs:

- General tab.
- 3D tab.
- Surfaces tab.
- Drawings tab.

General tab

In this tab you can configure displaying of the following objects on the Map, by selecting the appropriate check boxes:

For **Points**:

- The points' **Names**, **Codes**, **Notes, Icons**, and **Heights** along with the points.
- The **Auto Topo** and **Scanned** points.

For **Lines**:

- The **Stations**. Layout of the stations is defined by the Disp Dir As field.

For **Bing Maps**:

The check box is available when:

- GPS type of instrument is selected to work.

or

- Optical type of instrument is selected to work and a projection (not <none>) is selected.

If the check box is selected, after opening the Map the program will automatically load the Bing Maps for the current job objects from the Internet.

When the **Current Position in Main Map** check box is selected the program automatically snaps back to the center if it moves off the edge of the map.

3D tab

In this tab you can select the fill type and the way of moving and rotation of the job object(s) on the Map.

- Any surface in the 3D can be displayed as a **solid** model or as a **wireframe** model. To set the desired model, click the corresponding radio button in the **Fill mode** group.

In the **Rotation and moving** group you can select a way of objects moving and rotation :

- If the **Scroll 3D** radio button is selected, the objects will be moved in vertical or horizontal direction without any rotation.
- If the **Free rotation** radio button is selected, you can arbitrary rotate the job object(s) relative to some point. We recommend to use this way for low density of an arrangement of objects.
- If the **Free rotation about orbit** radio button is selected, you can rotate the job object(s) about orbit relative to some point. We recommend to use this way for high density of objects arrangement.
- If the **Rotation with fixed axis** radio button is selected, you can rotate the job object(s) about orbit relative to some vertical or horizontal axis. We recommend to use this way for high density of objects arrangement.

If the **Show objects border** check box is selected, you see a rectangle covering all objects of the job.

Surfaces tab

The left panel of the tab contains the list of the surfaces. You can highlight any surface and the right panel will show this object.

To show/hide surface(s) on the Map, select /unselect its checkbox.

Drawings tab

The left panel of the dialog contains the list of the imported vector images. You can highlight any file and the right panel will show this drawing. To show/hide a drawing on the Map, select /unselect its checkbox.

Objects near selected point

Typically, you can directly tap objects on the Map to highlight them. Alternatively, the dialog appears to select a desired object from the Objects near selected point. To do this:

1. Highlight the object in the list.
2. Click  to return to the Main Map with the marked object.

Point Near Selected Point

Displays a list of points that are close to the selected point and their point information.

Selected Point

Displays the Point Name, Code and Coordinates of the selected point.

Selected Line

The Selected Line dialog displays the name, codes, strings and length of the selected line.

Alignment Element Info

This dialog displays the information related to the selected alignment element.

Map

The Map icon on the Home screen opens the main Map. The main Map displays the map of the current job. To move the map display, hold down and slide the stylus on the screen. The Map maintains the scale after changing the status of the main map.

The basic commands of the main Map are available through:

- The map view tools which are arranged in two groups. More...
- The pop-up menus which depend on the objects selected. More...
- The drawing toolbar which allows you to create an object. More...
- The snap toolbar which allows you to create a point for the object which was selected in the Drawing Toolbar. More...

Map View Tools

The tool bar consists of two groups that may be opened/hidden by using the arrow icon.

Click a tool icon to execute the command:

Zoom In
Zooms the plot inwards.

Zoom Out
Zooms the plot outwards.

Zoom Window
Selects an area to center on. You may draw an area from low-right to top-left to highlight the object(s) you want.

Zoom All
Displays all objects in the map.

Center to Point
Selects a point for centering the plot.

Layers
Opens the Layers dialog.

Map Properties

Displays the map properties. The Map Properties are also available from every other pop-up menu.

3D View

Displays 3D view of the map. Then turns into the 2D View icon.

Drawing and Snap toolbars

Using the icons from the toolbars, you can create a point, a polyline segment, or an area.

Drawing toolbar

To open the drawing toolbar, click the button in the top left corner of the plan.

Point

Creates a point.

Polyline

Creates a polyline

Area

Creates an enclosed area.

Fillet

Creates a fillet for two lines.

Fit arc.

Creates an arc as the best fit to the suggested points.

Fit polyline

Creates a polyline as the best fit to the suggested points.

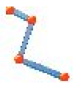
Point

The **Point** button allows you to add a new point to the existing entities in the selected snap mode. For more information about snap modes, see Snap Toolbar.

To create a point:

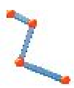
1. Click .
2. Select the required snap mode, by clicking its icon.
3. Click the required place on the map.

   The point is created according to the current snap mode.

 Polyline

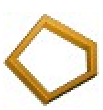
The **Polyline** button allows you to add a new polyline, by continuously creating the node points of the polyline when the snap mode is selected or not. For more information about snap modes, see Snap Toolbar .

To create a polyline:

1. Click .
2. Select the required snap mode, by clicking its icon.
3. Continuously click the required places on the map to draw a line. Each point will be created according to the current snap mode.

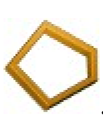
   Note: You may change the snap mode during the line creation. To do it, simply click the icon for the required snap mode.

   The polyline is created.

 Area

The **Area**button allows you to add a new enclosed area, by continuously creating the node points of the area in the selected snap mode. For more information about snap modes, see Snap Toolbar.

To create an enclosed area:

1. Click .
2. Select the desired snap mode, by clicking its icon.

3. Continuously click the required places on the map to create the nodes for the area. Each point will be created according to the current snap mode.

   Note: You may change the snap mode during the area creation by simply clicking another snap icon.

   The enclosed area is created.

 **Fillet**

The **Fillet** button allows you to add a fillet with the defined radius between two existing polylines/arcs.

Note: A fillet will be created as the arc, from the first selected polyline/arc to the second polyline/arc, clockwise.

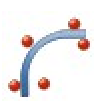
To create a fillet:

1. Click .

   The editbox for defining the fillet radius is displayed.

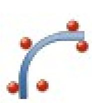
2. In the editbox, type the required radius of the fillet.
3. On the map, click the first polyline/arc.
4. On the map, click the second polyline/arc.

   The fillet is created from the end point of the existing entities.

 **Best fit arc**

The **Best Fit arc** button allows you to add a new arc as the best fit through the selected points.

To create a Best fit arc:

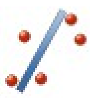
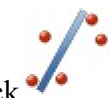
1. Click .
2. Select the points on the map.

3. After selecting points, click  . The new arc is created.

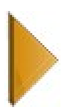
 **Best fit line**

The **Best fit line** button allows you to add a new line as the best fit through the selected points.

To create a Best fit line:

1. Click  .
2. Select the point on the map.
3. After selecting the points, click  . The new line is created.

Snap Toolbar

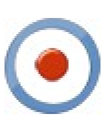
To open the snap toolbar, click the  button in the top left corner of the plan and select either point, or polyline, or area in the Drawing Toolbar. Using snap toolbar you can create points, lines or area as selected in the Drawing Toolbar.

End point snap mode

Creates either point at the end of segment, or segment /area using the end points of the polyline segment.

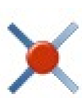
Mid-point snap mode

Creates either a point at the center of a segment, or segment /area using the middle point of the polyline segment.



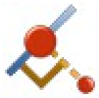
Circle center snap mode

Create a point in the center of arcs.



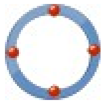
Line intersection snap mode

Create a point at the intersection of two lines.



[Perpendicular snap mode](#)

Creates a polyline perpendicular to an existing polyline.



[Circle quadrant snap mode](#)

Creates points at the circle quadrants.

End point snap mode

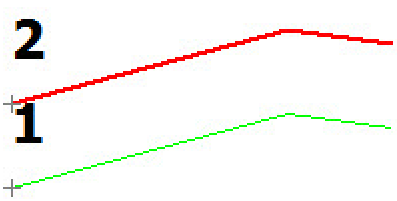
Use this mode to create:

- an end point for the selected segment of the polyline,
- a polyline segment between two end points,
- a closed area with the end points of the selected segment,
- a best fit arc with the end point of the selected segment,
- a best fit line with the end point of the selected segment.

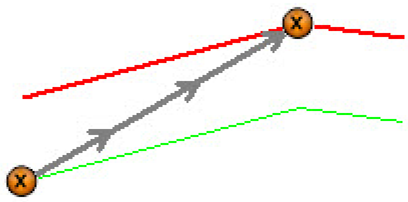
To use this mode:

1. Select the drawing tool: Point, Polyline, Area , Best fit arc, or Best fit line in the Drawing Toolbar.
2. On the snap toolbar, click  .
3. Click on a segment to create the end point on the nearest side.

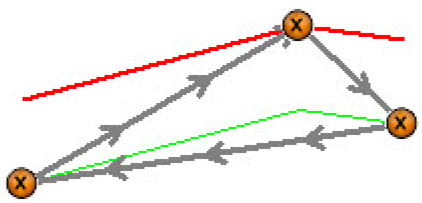
   If the Point button is selected, the software creates an end point for each segment that has been selected:



   If the Polyline button is selected, the software creates a segment between the end points of the segments that have been selected:



If the Area button is selected, the software creates an closed area using the end points of the segments that have been selected:

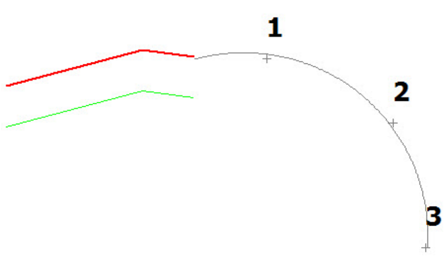


If the Best fit arc button is selected, the software creates:

- an end point for the segment that has been clicked

  and after deselecting 

- a new arc as the best fit through the points selected:

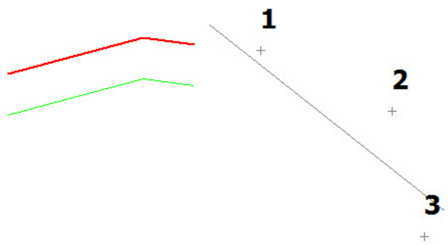


If the Best fit line button is selected, the software creates:

- an end point for the segment that has been clicked

  and after deselecting 

- a new polyline segment as the best fit through the points selected:

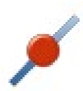


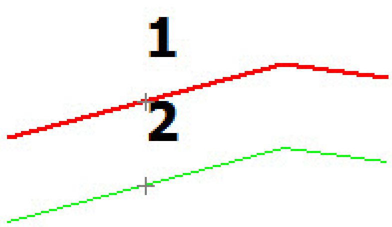
Mid-point snap mode

Use this mode to create:

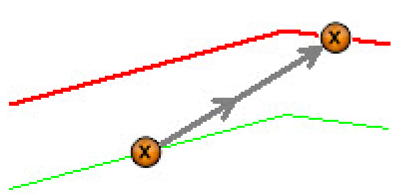
- a mid point for the selected segment of the polyline,
- a polyline segment between two (or more) mid points,
- a closed area with the mid points of the selected segment,
- a best fit arc with the mid point of the selected segment,
- a best fit line with the mid point of the selected segment.

To use this mode:

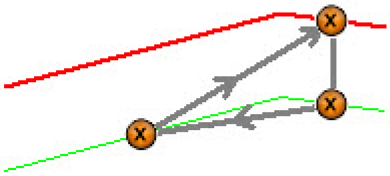
1. Select the drawing tool: Point, Polyline, Area, Best fit arc, or Best fit line in the Drawing Toolbar.
2. On the snap toolbar, click  .
3. Click on a segment to create the mid point.
   If the Point button is selected, the software creates a mid point for each segment that has been selected:

   If the Polyline button is selected, the software creates a segment between the mid points of the segments that have been selected:

   If the Area button is selected, the software creates an closure area using the mid points of the segments that have been selected:

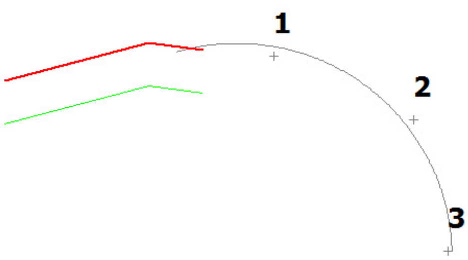


If the Best fit arc button is selected, the software creates:

- the mid point of the segment that has been selected

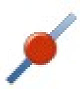
  and after deselecting 

- a new arc as the best fit through the point and other points of the job:

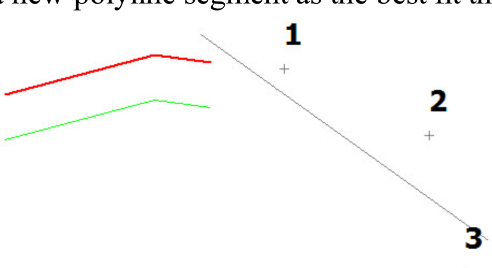


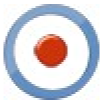
If the Best fit line button is selected, the software creates:

- an mid point of the segment that has been selected

  and after deselecting 

- a new polyline segment as the best fit through the points selected:



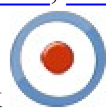


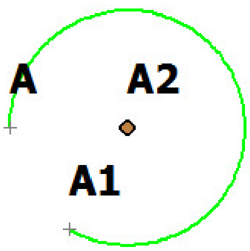
Circle center snap mode

Use this mode to create:

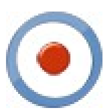
- a point at the center of an arc,
- a polyline segment from a circle center point to point or segment,
- a closed area from circle center point,
- a best fit arc with the circle center point,
- a best fit line with the circle center point.

To use this mode:

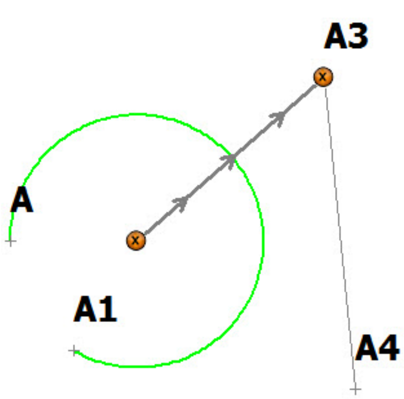
1. Select the drawing tool: Point, Polyline, Area, Best fit arc, or Best fit line in the Drawing Toolbar.
2. On the snap toolbar, click  .
3. Click on an arc or circle to create the center point.
   If the Point button is selected, the software creates an center point for the arc that has been clicked:



   If the Polyline is selected, the software creates:
   - the center point for the arc that has been selected

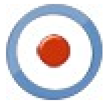
     and after deselecting 

   - a segment from the center point to any point or any segment that has been clicked:

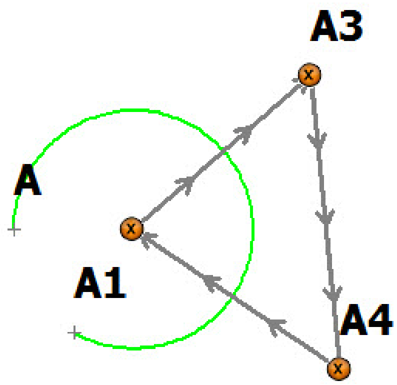


If the Area button is selected, the software creates:

- the center point for the arc that has been selected

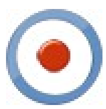
  and after deselecting 

- a closed area from this point and other points or segment that has been clicked:

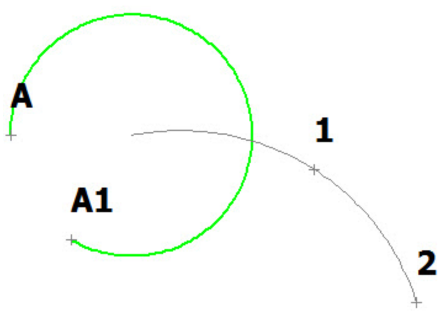


If the Best fit arc button is selected, the software creates:

- the center point for the arc that has been selected

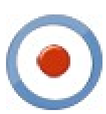
  and after deselecting 

- a new arc as the best fit through the center point and other selected points:

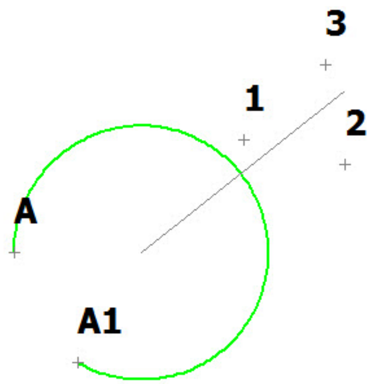


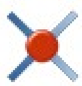
If the Best fit line button is selected, the software creates:

- a center point for the arc that has been selected

  and after deselecting 

- a new line as the best fit through the center point and other selected points:

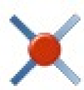


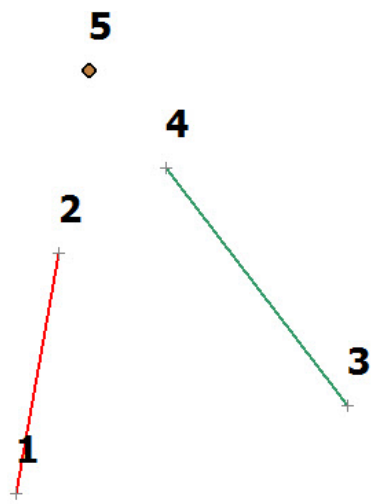
 **Line intersection snap mode**

Use this mode to create:

- a point at the intersection of a polyline and/or arc,
- a polyline segment from the intersection point to either a point or segment,
- a closed area from the intersection point,
- a best fit arc with the intersection point,
- a best fit line with the intersection point.

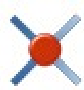
To use this mode:

1. Select the drawing tool: Point, Polyline, Area, Best fit arc, or Best fit line in the Drawing Toolbar.
2. On the snap toolbar, click  .
3. Click a polyline/arc and the intersecting polyline/area to create intersecting point:
   If the Point button is selected, the software creates an intersection point of the two line segments that have been selected:

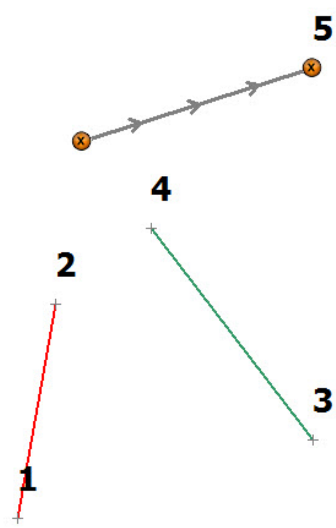


If the Polyline button is selected, the software creates:

- a point at the intersection of the two line segments

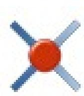
  and after deselecting 

- a line segment from the point to any selected point or line segment:

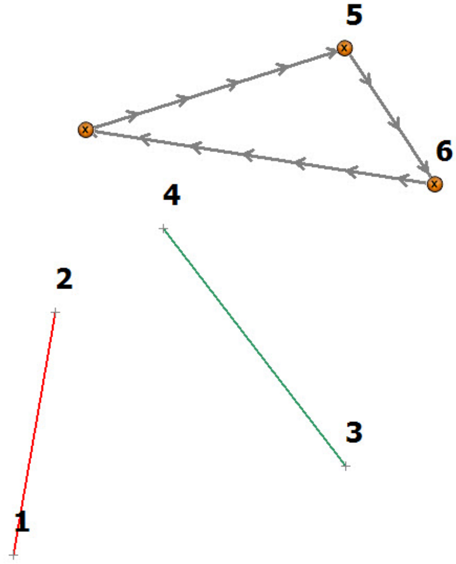


If the Area button is selected, the software creates:

- a point at the intersection of the two line segments

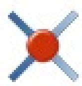
  and after deselecting 

- a closed area from the point and other selected points or line segments:

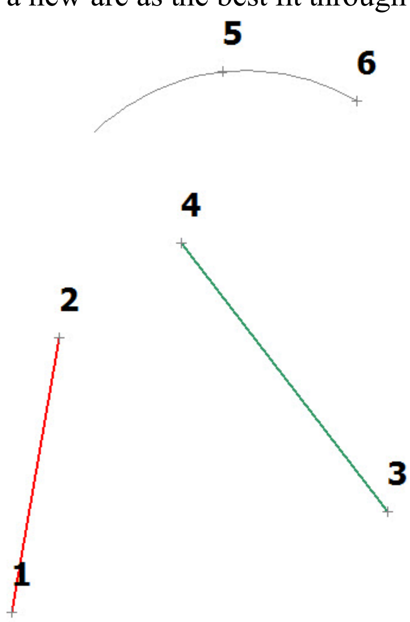


If the Best fit arc button is selected, the software creates:

- a point at the intersection of the two line segments

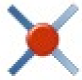
  and after deselecting 

- a new arc as the best fit through the point and other selected points:

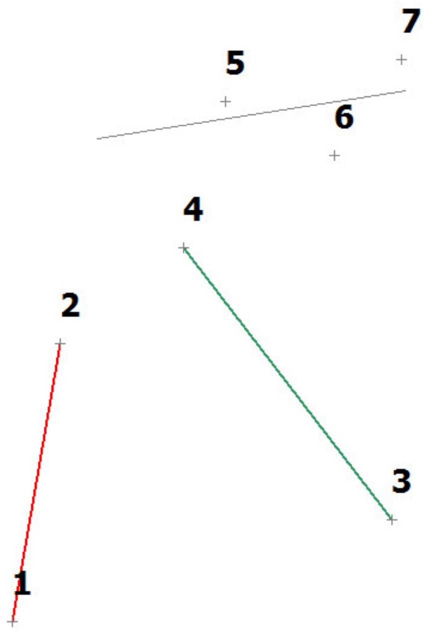


If the Best fit line button is selected, the software creates:

- a point at the intersection of the two line segments

  and after deselecting 

- a new line as best fit through the point and other selected points:



 **Perpendicular snap mode**

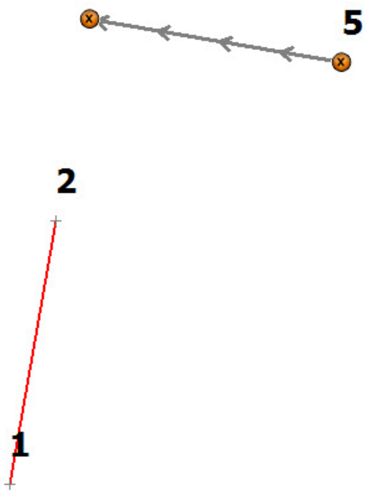
Use this mode to create a line segment perpendicular to an existing polyline.

Note: Before using this snap mode create at least one polyline segment.

To use this mode:

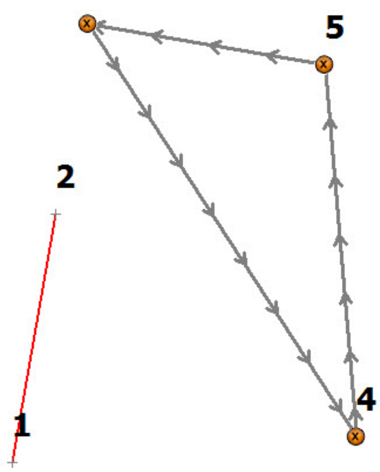
1. Select the drawing tool: [Polyline]() or [Area.]()
2. On the snap toolbar, click .
3. Click a polyline/arc from which to create the perpendicular line.
   If the [Polyline]() button is selected, the software creates a perpendicular line from a selected point or polyline to

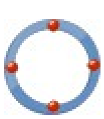
the selected segment:



If the [Area](#) button is selected, the software creates:

- a perpendicular line from a selected point or a polyline to the line segment that has been clicked and after deselecting 
- a closed area from the intersection point, the perpendicular line and other selected point(s):



 Circle quadrant snap mode

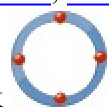
Use this mode to create:

- an point at the intersection of the circle quadrant axe with its circumference,
- a polyline segment between two (and more) intersection points,

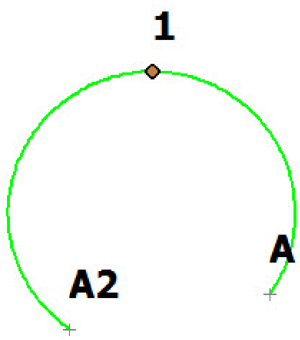
- a closure area from the intersection points,
- a best fit arc with the intersection point,
- a best fit line with the intersection point.

Note: The nearest to the click point intersection will be used.

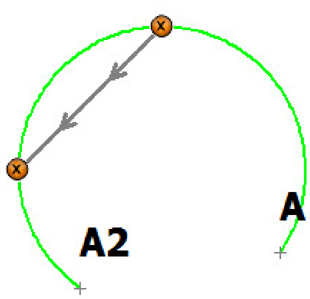
To use this mode:

1. Select the drawing tool: Point, Polyline, Area, Best fit arc, or Best fit line in the Drawing Toolbar.
2. On the snap toolbar, click  .
3. Click on a circle or arc to create the quadrant point:

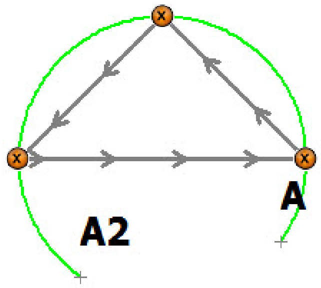
   If the Point button is selected, the software creates a point at the intersection of the circle quadrant axes with its selected circumference:

   If the Polyline button is selected, the software creates a polyline segment between two (or more) intersection points:

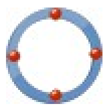
   

If the Area button is selected, the software creates a closed area from the intersection points:

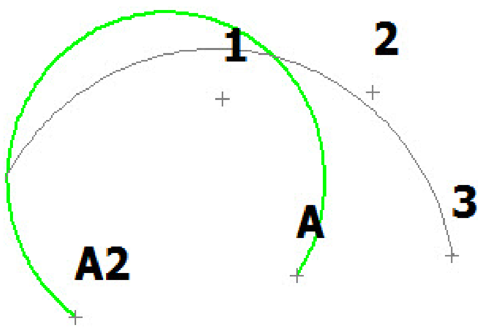


If the Best fit arc button is selected, the software creates:

- point(s) at the intersection of the circle quadrant axes with its circumference

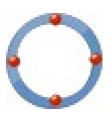
  and after deselecting 

- a new arc as the best fit through the point(s) and other points of the job:

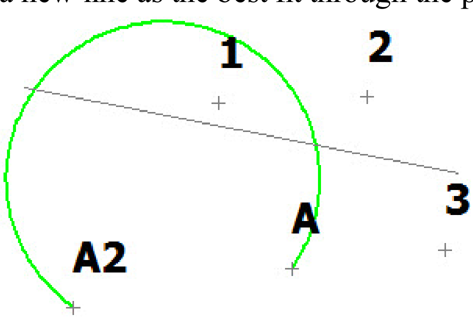


If the Best fit line button is selected, the software creates:

- point(s) at the intersection of the circle quadrant axes with its circumference

  and after deselecting 

- a new line as the best fit through the point(s) and other points of the job:



Connections

This dialog allows you to perform the following tasks:

1. To connect with the device of the selected type and select a configuration style to work at the site.
   Find out more on Device connection
2. To enable a connection prompt upon starting the job.
   Find out more on Device connection
3. To connect with MAGNET Enterprise web-server and the project for data exchange.
   Find out more on Enterprise Connection
4. To connect with the network server.
   Find out more on Network connection
5. To re-connect with a HiPer SR base.
   Find out more on LongLINK connection
6. To connect with SiteLINK 3D server.
   Find out more on SiteLINK 3D connection

Connecting with Device

The General tab of the Connections dialog allows you to change the type of the device to work and to select a job configuration before connecting to the device.

- Select the **GPS** type of instrument and the configuration to work in the current job. Select the **Base** or **Rover** radio button when working with the base receiver or the rover receiver in RTK surveys, respectively.
- Select the **Optical** type of instrument and the configuration to work in the current job.
- When **Connect to the last used BT device** is selected (by default), automatically connects to the last used Bluetooth device. Find out more on Bluetooth connection. If you clear this check box, the device discovery will be initiated by pressing the Connect button.
- Clearing the **Prompt at startup** check box will prevent the program from connecting to an instrument until you click the icon  on the Home screen or enter a Survey or Stakeout dialog. However, the Network settings will be available.
- Clicking the **Connect** button initiates the connection to the device associated with the currently selected configuration
- Clicking the **Disconnect** button closes the connection with the device.

Bluetooth Connection

The procedure of Bluetooth connection includes three stages:

Device Discovery
Authentication
Connection to Bluetooth device

Device Discovery

The *Select Bluetooth Device* dialog lists all visible Bluetooth Devices.

- Each device's *Name* is displayed. If the name cannot be resolved, Bluetooth Unique Address is displayed instead. Type of device and its services are displayed if the device provides this information.
- The dialog title displays the type of device that application tries to connect to.
- If a device is not present in the list or some devices did not report their names in a timely manner, try refreshing the list using the **Refresh** button. If the device is still not listed, then the distance to it can be too large or the device can be connected to a different controller. Connection with the device is possible even if its name was not resolved and only Bluetooth Unique Address is listed.
- Pressing the **Select** button will initiate authentication with device.

- If Bluetooth hardware is disabled (powered off) or the controller has an unsupported Bluetooth stack, a warning will be displayed. Currently only the Microsoft Bluetooth stack is supported.

Authentication

The *Bluetooth PIN* dialog displays information about the selected device.

- The application maintains authentication information independently of Bluetooth UI in the Operating System. There is no need to pair devices beforehand, the application will perform the pairing.
- The application supports connections with devices that do not require a PIN code. To use this feature, clear the **Require PIN** check box. If the PIN code is provided and a connection is successful, the application will store the PIN code in a protected area for future use.
- Press the **Connect** button to initiate a connection.
- If a connection succeeds, Bluetooth Unique Address and its authentication will be stored. Next time the application will not display the Device Discovery.

Connection to Bluetooth device

When the application tries to establish a connection with Bluetooth port, it displays the *Accessing Bluetooth Device* dialog. The title of the dialog displays the type of device that application tries to connect to.

- If a connection cannot be established in a timely manner, the dialog will automatically stop the connection. A connection can be canceled manually and with the **Cancel** button.
- It is possible to initiate the Device Discovery procedure using the **Change Device** button. Also, the device discovery can be initiated by clearing the **Connect to the last BT device** check box before connecting.

Connecting with Enterprise

The Enterprise tab allows you to establish a connection with an enterprise project:

1. Enter your **Login** information for connection to the Magnet Enterprise server.
2. Select a project from the **Connect to project** drop-down list. You can create a new project using the nearby icon.
3. If required, select the **Connect on startup** check box to connect to the project when running the MAGNET Field Site.
4. Click **Connect** to establish the connection. Connection status displays the result.

Connecting with Network

The Network tab displays after you connect to a GPS device with Internal CDMA and GPRS modem for a Network survey. This tab visualizes the actions the program performs to connect the modem to the Network server.

- The action list is a read-only control, that shows available actions. Current action is marked with a check (last in the list).
- The **Mount Point** box shows an available list of mount points. To refresh the list, click . Click  to view information on the highlighted mount point.
- The **Signal** indicator shows the signal strength. More bars at indicator means better radio signal. This indicator is enabled only if the Check signal quality is enabled in Miscellaneous settings.
- The status field shows the status of the current operation (that is a subpart of an action).
- The connection (disconnection) process can be initiated automatically if an auto-connection (disconnection) to Network server is enabled in Miscellaneous settings or manually by pressing the **Connect** (**Disconnect**) button.

Mount Point Information

This dialog displays the information related to the selected Mount point.

Information displayed includes: type, mountpoint, identifier, format, format-details, carrier, nav-system, network, country, latitude, longitude, nmea, solution, generator, compr-encrypt, authentication, fee, and bitrate.

LongLINK Connection

When you are working with a HiPer SR rover, a LongLINK connection with a HiPer SR base is automatically performed if the rover finds a single base. The LongLINK tab allows you to select a base to re-connect if there are several bases found. The tab displays:

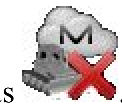
- The connected base transmitting corrections which is marked with the icon .
- All other available bases with the icon .
- The base data that contain the bases' names, the site IDs, coordinates, the number of the available bases and the level of the signal power in percentage.

The buttons serve the following purposes:

- **Disconnect** breaks the current connection.
- **Refresh** renews the list of available bases.
- **Connect** initiates the connection with the selected base. The Connect button changes for Disconnect.

Connecting with SiteLINK 3D

The SiteLINK 3D tab allows you to establish a connection with SiteLINK 3D server:

1. Enter appropriate login information for connection: server IP, port, password and username.
2. If required, select the **Connect on startup** check box to connect to the server on starting the MAGNET Field Site.
3. Click **Connect** to establish the connection.
4. Connection status displays the result. The icon  indicates the successful connection.
5. Click **Details** if you want to learn more on the failed connection status .

Setup folder

Before conducting the survey you may need to perform some preliminary work that depends upon the current job configuration.

Follow a link to learn more:

Setup GPS survey

Setup Optical (Robotic) survey

Setup GPS folder

Click an icon to perform the task:

Start Base
Sets the Base Receiver in RTK survey. Available after you establish a connection with the base receiver.

Connect
Switches between GPS and optical instruments, and sets connections with the instrument, a network, and with MAGNET Enterprise.

Localize
Calculates localization parameters of the mathematical coordinate transformation between an original coordinate system, in which the job points are measured or given, and a local coordinate system, in which control points are known. Find out more...

GPS Status
Displays information about the current position of the GNSS receiver, RTK status, and the satellite constellation.

mmGPS Init
Sets up mmGPS+ system for RTK surveying. Available after you select the mmGPS+ system in peripherals for the rover receiver configuration.

Simulator
Sets initial WGS84 position for GPS simulation configured.

Status

This allows you to check the status of a GPS+ survey. The Status dialog contains information about the current position of the receiver, RTK status, and the satellite constellation.

- Position
- System
- Log History
- Multi Base Status
- Scatter Plot
- SVs

Position

The Position tab displays:

- Total number of the available satellites. The lock icon signifies the number of the satellites tracked, and the star icon shows the number of satellites used in position determination.
- Current UTC time.
- Coordinates of the position in the selected coordinate system and units.
- PDOP value; it is a factor depending solely on satellite geometry describing how the uncertainty in the coordinates will depend on the measurement errors. PDOP is proportional to the estimated position uncertainty.
- H and V stand for HRMS and VRMS, the RMS values of the horizontal and vertical coordinates, respectively.
- Base Dist: slope distance to base antenna. The field is empty if no differential corrections are received.

Settings icon opens the dialog where Elevation Mask or Base Make At Rover parameter can be changed.

Other pages in the Status dialog:

- System
- Log History
- Multi Base Status
- Plots
- SVs

The icon  opens a pop-up menu which varies depending on the configuration type used. More...

Settings

To set parameters for correct positioning in RTK surveys:

1. Select **Settings for** which receiver you want to apply: *Base* or *Rover*.
2. In **Elevation Mask**, enter the elevation angle below which the satellites will be rejected while positioning.
3. **Base Make at Rover** allows you to designate Base Make (IGS Class) used by the Rover receiver to account for GLONASS biases. This also allows you to override Base Make automatically detected by the Rover receiver if this information is transmitted by the Base.
4. Select the **Canopy environment** check box to instruct the RTK engine to use less rigid thresholds when filtering out measurement outliers. This mode is recommended when working under a tree canopy or in other cases of high multipath.

Note: If the receiver is connected, the settings are applied upon closing the dialog.

Status pop-up menu

The pop-up menu can contain a different set of the following options depending on the configuration type used

Rover Antenna Setup
Config Radio
Config Beacon
Config RE-S1 Repeater
Reset RTK or **Reset DGPS** - the command that reinitializes the receiver.
mmGPS+ Options
Mission Planning
Clear NVRAM - the command that resets the receiver parameters to factory default values (such as active antenna input, elevation mask and recording interval, and information about the receiver's internal file system). This command will not delete any files from the receiver memory. After clearing the NVRAM, the receiver will require some time (around 15 minutes) to collect new ephemerides and almanacs.

System

Open the System tab to view the information about the current state of the RTK measurements:

- *Position Type* - Indicates the solution type of the position.
- *Common Sats* - Number of common satellites between the base and rover.
- *Initialized Sats* -Number of satellites initialized.

- *Radio Link* -Quality of the radio link.
- *RTK Age* - Age of the last RTK message in seconds.
- *Receiver Memory (KB)* - Available memory in the receiver for storing TPS files.
- *Receiver Power(%)* - Percentage of receiver power remaining.
- *Controller Memory (KB)* - Available memory in the controller.
- *Controller Power(%)* - Percentage of controller power remaining.
- *NetRTK (MAC) Status* - Applies to Network RTK style with MAC corrections selected. Will display 'Yes' if MAC is used for position computation.
- *Base Make at Rover - Requires Topcon receiver f/w at rover 3.4 or higher.*
  If Automatic detection of Base Make is selected in style settings, this field will report the Base Make that was detected by the Rover receiver. If the Base receiver does not support IGS Class extensions to correction formats or if the required RTCM messages were not enabled on the base, this field will report a dash (-), and the rover will apply default GLONASS corrections. You can override this by pressing the Settings button in the Status dialog or in the Styles Settings (Advanced). No Base Make At Rover will be reported by MAGNET Field Site in this mode.

Other pages in the Status dialog:

Position
Log History
Multi Base Status
Plots
SVs

Solution type

Possible solution types:

- No Solution - receiver cannot produce a solution (not enough satellites or incorrect antenna model)
- Autonomous (Standalone)
- DGPS (Code-differential)
- Float
- Fixed (RTK)

mmGPS+ solution marker indicates that solution was produced using mmGPS+ technology

---

Log History

If the Log History is available, a Log History tab is displayed.

The Log History tab graphically displays the usage of satellites over time. For convenience, the field is divided into 5-minute portions along dotted lines. The starting time and next half hour is marked with time labels.

Other pages in the Status dialog:

- Position
- System
- Multi Base Status
- Plots
- SVs

Multi Base Status

The Multi Base Status tab displays the information about the current state of the RTK measurements in the multiple base configuration.

RTK - Use: a checkmark indicates this base station is currently automatically used by the RTK engine.
RTK - ID: the numerical ID of this base station.
RTK - Base: the name of this base station. (Only available if this base is currently in use)
RTK - Age: the age of the last RTK message received by this base station.
RTK - Link: the quality of the radio link of this base station.
RTK - Type: Indicated the solution type of the position (only available if this base is currently in use)
RTK - Dist: the distance between this base station and the rover. (Only available if this base is currently in use)

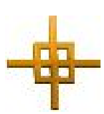
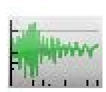
Other pages in the Status dialog:

- Position
- System
- Log History
- Plots
- SVs

Plots

The Plots tab displays the current receiver position changing in time: either the current receiver vertical position or the horizontal position relative to the position in a local (northing, easting) coordinate system.

The buttons on the plot are used:

- to switch between horizontal  and vertical  plots
- to zoom in  and out 

- to open properties 

Other pages in the Status dialog:

Position
System
Log History
Multi Base Status
SVs

Properties of Horizontal Plot

Show Grid

If selected, displays the local coordinate axes.

Auto Zoom

If selected, automatically scales the horizontal scatter plot to fit into the dialog.

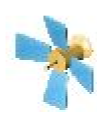
Properties of Vertical Plot

Time Window

Duration in seconds for the time axis.

SVs

The SVs tab displays a graphical representation of the position of the satellites on the sky or the S/N ratios.

- Clear the **GPS+SBAS** check box to hide the GPS and SBAS satellites. These satellites are marked with  icon.
- Clear the **GLNS** check box to hide the GLONASS satellites. GLONASS satellites are marked with  icon.
- Toggle between **SNR** and **Plot** to observe the skyplot or the signal-to-noise ratio for the satellites.
- Click **List** to obtain the table displaying the satellites parameters:
  PRN: shows the number of the satellite
  H/U: shows whether the satellite is healthy or unhealthy
  EL: shows the elevation angle of the satellite

AZ: shows the azimuth of the satellite

SNR1: L1 signal to noise ratio

SNR2: L2 signal to noise ratio

USED: shows whether the satellite is used in position computation. You can select whether a satellite should be used or not by selecting the corresponding row for the satellite in the list and clicking on the USED column header. This will toggle the use of the satellite.

Other pages in the Status dialog:

Position
System
Log History
Multi Base Status
Plots

Mission Planning

The Mission Planning option enables you to examine the expected observation conditions (such as satellite visibility and PDOP). This can be done in order to find out the best timing for observations at the specified point.

To perform mission planning:

1. Select *Setup* / *Status* and the *Mission Planning* option from the pop-up menu.
2. Configure settings for observations in the *Mission Planning Setting* dialog. More...
3. Once the Mission Planning Setting dialog is closed the Mission Planning dialog will calculate and show the position of satellites in the first ten minutes of the specified time on the *Sky Plot* (the celestial sphere).
4. The *Local* field displays the date and time (local time) of the calculation result.
5. The dialog also provides information about the given position as applied to the calculations, the *PDOP* value, and the number of *GPS* and *GLONASS*.
6. If required, you can hide all the GPS or GLONASS satellites from the view. To do this, uncheck the corresponding box.
7. Use the appropriate button to calculate and show the satellites in every ten minutes of the specified time:

    > or < to move forward or backward, respectively
    I< to return to the start time
    II to pause at any ten-minute interval

8. To view different graphical satellite information, select a desired option from the drop-down selection list:

Sky Plot
Sat List
Sat Num
PDOP

Mission Planning Setting

To configure settings for Mission Planning:

1. Enter the coordinates of the current position in one of the two possible ways
    - automatically by clicking 
    - manually in the current coordinate system after clicking 
2. Select the *Date* and *Start* and *End Time* of observations as required.
3. If required, change the Elevation Mask from the default 15 degrees for a desired value.

Satellite List

The Satellite List option allows you to view graphically the availability of either GPS or GLONASS satellites changing in the specified time.

Satellite Number

The Satellite Number option allows you to view graphically the total number of GPS and GLONASS satellites changing in the specified time.

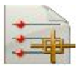
PDOP

The PDOP option allows you to view graphically the PDOP value changing in the specified time.

 Start Base (RTK)

The Start Base dialog contains information about the Base receiver and is used to set up the Base.

To set up the Base:

1. Enter the name of the **Point** where the Base receiver is located. It can be chosen from the map  or the list  of the job points, or can be entered manually.
2. Select the **Code** for the point from the drop-down list. If required, set a string. Click  to set the point's attributes. More...
3. The coordinates of the base point in the selected coordinate system will be shown.
4. You can enter the measured coordinates of the current point. To measure the current position:
   - Enter the antenna height and set the type of height measurement (vertical or slant). To do this, click the button and select *Edit* from the drop-down menu. The menu retains your settings. You can edit the antenna type, the value of the antenna height, and the type of height in the Antenna Setup dialog.
   - Click the button  . Once pressed, the button turns into  .
   - Click this button to stop logging the position. The average measured coordinates will be displayed. The Pos field appears to show the number of the measurements used for averaging.
5. Click the **Start Base** button to start the receiver as the Base transmitting the correction data.
   Note: If you start the base with a UHF radio and only for the first time in the job, clicking the Start Base button initiates the Quick Radio configuration to quickly set up the UHF radio for data transmission. Learn more about how MAGNET Field Site controls Quick Radio.
6. The **Duration** field displays the length of time you want the receiver to log data to the raw GPS data file for the Post-Processed RTK. The logging data will start by the Start Base button and stop by the Stop Base button.

The icon  opens a pop-up menu of the following options. More...

Antenna Setup

To set up the GPS antenna:

1. Select the model of the Topcon **Antenna** if available for selection (for instance, HiPer V, GR-5, GR-3).
2. In the Height panel you can view/edit:
   - the value of the antenna height,
   - the method used to measure the antenna height:
     - Vertical - measured from the ground point to the antenna reference point (ARP) located on the bottom of the receiver.
     - Slant - measured from the ground point to the antenna slant height measure mark (SHMM).
   - the value of the adapter height (adapter - an additional device which can be set between a GPS receiver and a rod).
3. For the external GNSS antenna you can enter its Serial Number.
4. Click to save the settings.

Quick Radio

The Quick Radio functionality enables quick configuration of a UHF radio channel for data transmission.

These are some details about Quick Radio work:

- If you change a radio parameter value using any other application, MAGNET Field Site will not consider the new value, and Quick Radio will not work correctly.
- If you connect the same base receiver as the RTK / Network RTK rover, and then back as the base, Quick Radio will be reset and will work again when you start the base.
- If you change the power using the Config Base Radio option, Quick Radio will be reset and will work again when you start the base.
- If you change the channel using either the Config Radio dialog or the Quick Radio configuration during the last Start Base, Quick Radio will not be reset and will not work when you start the base.

Start Base pop-up menu

The pop-up menu can contain the following options:

Status
Config Radio depends on the configuration for the survey
Multi Base available for RTK with CMR+ format of correction data
Grid to Ground available if starting the base in a grid coordinate system

---

Multi Base

For Multi Base mode in RTK survey, all Base receivers must be configured to transmit at the same frequency and must transmit CMR+ format corrections. The Rover receiver must be configured to receive only CMR+ messages.

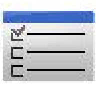
To set multi base mode for surveying:

1. From **Base Station ID**, select the identifying number of the base, which is sent as part of the CMR+ messages. Each base in a network must have a unique ID.
2. From **Transmit Delay**, select the signal transmission delay from the current base. This parameter is set to ensure that the radio signals from multiple bases do not overlap. Each base should be set at least 250 milliseconds apart from any other. (At least 500 milliseconds for data transfer rates less than 9600 bps.)
3. Select the **Use Multi Base** check box to enable multi base mode.
4. Click  to save the settings and to return to the Start Base dialog.

Localization

Localization is used for transformation of the coordinates between a local (ground) system and other (WGS84) system, in which the job points are measured. To perform localization, you need some control point(s) which have independent coordinates in both systems: local ("Known Point") and job coordinate system ("Measured Point").

1. Select the **Type** of Localization (by the job coordinate system). The WGS84 selection is always available. Datum or Grid selections are available only if you selected a specific coordinate system in the Coordinate System dialog before visiting the Localization dialog. On Localization dialog it is possible to change the Localization type after control points were added.
2. Click **Add** to open the Localization Point dialog for adding control points. Once a points pair is added, the localization is computed. Each point has a level of reliability specified with the values of the residuals along the horizontal and the vertical axes and the Control parameters, that shows the status of the point.
   Note: The localization is recomputed every time a new point pair is added to the list of localization points. The new coordinate system will be saved under the name "Localization" and is automatically selected.
3. Select the **Keep scale 1.000000000** check box to preserve the localization from a scale transformation as required.
4. If required, click **Edit** to open the Localization Point dialog to change the control point added.
5. Click **Remove** to remove the highlighted points.
6. Click **Details** to view the results of the localization.

Click  to change Settings for the survey if necessary.

Related tasks:

- Basic Concept of Localization
- Localization with Stereographic Projection on WGS84
- Localization with Stereographic Projection on any Datum
- Localization with any Predefined or Created Projection
- One-point Localization
- Two-point Localization
- Three-point Localization
- Performing Localization
- Importing Localization Data to Other Job

Add Point

To add points to localization:

1. Select the **Use Horizontal** and **Use Vertical** check boxes to use these points for horizontal and/or vertical localization. The horizontal and vertical use of any control point can be changed by highlighting in the Localization dialog and then tapping on the header of the H Control or V Control. This toggles the display between "yes" and "no".
2. In the **Known Point** field, enter the *Point* in the local (ground) coordinate system. You can enter the point manually or select it from the map or from the list of the job points.
3. In the **Measured Point** field, enter the *Point* in the coordinate system that is set for the current job. When computing localization parameters these coordinates will be transformed to the local system. The type of transformation can be selected in Localization dialog.

   You can enter the point manually, select it from the map or from the list of the job points, or set the control point to the current location. Select the Code for this point and click to set the point's attributes . More...
4. Click **Start Meas** to use the current location. The # field shows the number of the accepted epochs. The parameters of the logging are set through the Settings button.
   If a point with the same name already exists, the application will open the Point Check notification dialog. You can overwrite, rename, or store the point as a check point.
5. Click to save the point and return to the Localization dialog with a newly added point.

Localization Results

The Localization Results dialog contains the calculated parameters of the localization: the global coordinates (Lat0, Lon0, h0), the corresponding local coordinates (N0, E0, Elev0), the scale parameter (Scale), the rotation parameter (Az) and the plane slope angles corresponding to north and east directions.

Simulator

Allows you to setup the initial WGS84 position for GPS simulation. You can enter the position manually or select a point from the map or from the list.

If simulation moving speed is not zero, the current position starts to drift immediately. The current position is remembered between MAGNET Field Site runs.

Moving speed and direction can be changed in the main map or in the Topo dialog using arrows.

---

mmGPS Initialization

The procedure of initialization of mmGPS+ systems includes two steps:

1. Connect the controller and transmitter. Calibrate the transmitter with the correct channel and communication port and also setup the transmitter's height and locate it at the jobsite. Learn more about that from Transmitter Data and Transmitter Position.
2. Unplug the controller from the transmitter. Connect the controller and GPS receiver to initialize the Sensor.

Click  to display additional functions:

- *Field Calibration*: Opens the Calibration dialog to ensure correct grade in the self-leveling mechanism of the transmitter.
- *Known Point Offset*: Opens the Known Point Offset dialog to check the results of a resection and optionally to adjust the transmitter's height using the new offset.
- *Advanced Sensor Options*: Opens the Advanced Options dialog.

Transmitter Data

The Data tab allows calibration of the transmitter with the correct channel and communication port:

- The transmitter list contains the following items:
  Name: The name of the transmitter.
  ID: The ID that corresponds to the channel of the transmitter.
  Data: The status of calibration data.
- Click **Add** to add a transmitter to the transmitter list. More...
- Click **Edit** to change the information on the existing transmitter if required.
- Click **Delete** to remove the highlighted transmitter from the list.

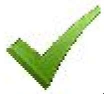
Transmitter

To add a connected transmitter:

1. Enter the **Name** of the transmitter.
2. From **Com Port**, select the communication port of the controller that is used for connection with the transmitter.
3. Click **Get Data** to retrieve the transmitter's data.
   The *ID* that corresponds to the channel of the transmitter

The status of *Calibration Data*

The *Firmware Version*

4. Click **Clear Data** to clear the data fields if required.
5. Click  .

Transmitter Position

To set up the transmitter's height and location at the jobsite:

- The transmitter list contains the following items:
  Name: The name of the transmitter.
  ID: The channel of the transmitter.
  Point: The point over which the transmitter is setup.
- Click **Resect** to perform resection for an unknown transmitter location in the event that a point has been lost. More...
  Note: This operation requires that the transmitter and sensor have already been setup.
- Click **Edit** to enter the transmitter's position. More...
- Click **Delete** to remove the transmitter from the list.

Resect mmGPS+

To measure an unknown transmitter location using the rover and three or more points:

1. With the controller and sensor connected, set up the sensor. More...
2. Perform resection measurements. More...
3. Observe calculated data. More...

Sensor

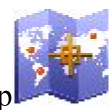
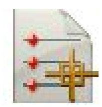
To set up the sensor for measurements:

1. Select the **Receiver Port** that connects the receiver and sensor.
2. From **Transmitter ID**, select the transmitter's channel. The ANY selection will allow the sensor to independently select the transmitter with the smallest error rate.
3. Select the **Sensor Gain** to set the sensitivity of the sensor to the transmitter's laser beam.
4. The **Firmware Version** for the sensor is shown.
5. If **Known Trans Horz Pos** is selected, the Known Point dialog displays. Select the point over which the

transmitter is setup.
6. Click **Init Sensor** to start initialization process.

Resect

To perform the measurements from the rover point to the point over which the transmitter is set:

1. Observe the information about the current state of measurement displays:
    - An icon for the transmitter's active beam .
    - The quality of the radio link.
    - The type of the position calculation method.
    - The RMS errors for horizontal and vertical coordinates.
    - The number of the satellites tracked and used in position calculation.
2. If using an unknown point, click .
3. If using a known point:
    - Enable the **Known Point** check box and select the point using the map  or list  buttons.
    - Enter the **Antenna Height** and set the type of height measurement (vertical or slant).
    - Click .
4. During the measurement, the *Logging* field displays the counter of the epochs collected.
5. The ***Meas*** field shows the number of measurement.
6. When the desired amount of epochs is achieved, click .
7. Move to the next point and repeat measurement steps for it.

Data

On this tab:

1. View the results of resection calculation. Data will display only after three or more points have been measured.
2. Click **Re-Meas** to re-measure a point if required.

3. If the resection values are acceptable, click **Accept** and view the point information for the transmitter.
4. Enter any other desired information and click to save the transmitter's point information.

Known Point

To setup the transmitter over a known point:

1. Select the **Point** the transmitter is installed over. It can be selected using the map or list buttons.
2. In the **Transmitter** area:
   - The transmitter's **Name** and channel **ID** will be displayed.
   - Select the **Fixed Tripod** check box if you use a tripod of a fixed height. Select the required value of the height from three available.
   - In **Ht**, enter the height of the transmitter: either *Vertical* to the *Base* of the transmitter or *Slant* to the cross *Mark* on the transmitter's side.
3. Click .

Sensor

To upload transmitter calibration information to the sensor and set up the sensor for receiving the transmitter's laser beam:

1. Select the **Receiver Port** that connects the receiver and sensor.
2. From **Transmitter ID**, select the transmitter's channel. The ANY selection will allow the sensor to independently select the transmitter with the smallest error rate.
3. Select the **Sensor Gain** to set the sensitivity of the sensor to the transmitter's laser beam.
4. The **Firmware Version** for the sensor is shown.
5. Click **Init Sensor** to start initialization process.

Field Calibration

The Field Calibration function fixes errors in incline in the self-leveling mechanism of the transmitter.

To perform field calibration:

1. Set the transmitter into calibration mode. To do this, hold the plumb beam key, then press and release the power key.
2. Connect the controller and sensor. Face the sensor towards the transmitter at the distance of several feet. Ensure that the sensor remains steady throughout the calibration process.
3. The **Transmitter Name** displays.
4. Click **Next**. A wizard will guide you through the calibration process. When the calibration completes, Update Calibration Data if required.

Update Calibration Data

The Update Calibration Data dialog displays after performing Field Calibration if it is determined that the transmitter leveling offsets need to be adjusted.

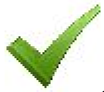
To update the calibration data:

1. Disconnect the controller from the sensor and connect with the transmitter.
2. Select the **Communication Port** that connects the controller and transmitter.
3. Click **Update Data** to send the offsets information to the transmitter. MAGNET Field Site uploads the calibration data to the transmitter and automatically turns off the transmitter.
4. When finished, close a successful message and initialize the sensor. More...

Known Point Offset

The Known Point Offset function is used to compute the height offset between the rover and a currently occupied known point. This can be used as an adjustment for the transmitter height.

1. The **Transmitter Name** field displays the name of the transmitter.
2. Select the rover's known **Point**. It can be selected using the map or list buttons.
3. Enter the **Antenna Height** and set the type of height measurement (vertical or slant).
4. The **Num Epochs** field displays the number of GPS epochs used in the measurement.
5. Click to start the measurement process. After pressing, the button changes to , and the counter of the epochs collected appears. You should wait until the epochs collected are averaged. Click to cancel.
6. When the averaging completes, the **Height Offset** will display the difference in height between the known measurement and the current rover measurement.

7. Click  , then **Yes** on the warning dialog to adjust the transmitter height using the offset. This offset will be automatically added to the transmitter's height.
8. When finished, initialize the sensor. More...

Advanced Sensor Options

In the mmGPS+ Options field you can:

1. Select the **Init Time Improvement** check box to improve the RTK fix time for the receiver.
2. The **Weighted Height** check box is selected by default to use the weighted height values from GPS and mmGPS+ measurements.

Topo

The Topo dialog enables you to conduct a stationary survey.

This dialog includes the following tabs:

- Measure contains the initial data for the survey and displays the progress of the survey.
- Data contains information about the stored point.
- Map allows you to perform measurements and shows the stored points graphically.
- Offsets allows you to set the offset point for the measurement.

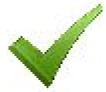
Click to open the pop-up menu of additional options. More...

Measure

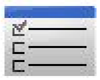
The Measure tab contains the initial data for the survey and displays the progress of the survey. The information about the receiver operation is shown. It duplicates the information on the Status dialog. More...

To perform measurements:

1. Enter the name of the **Point** to be logged.
2. Enter code and attribute information for the point.
   - You can select a **Code** from the drop-down list. Code needs to be defined at the time it is entered if it is not a code that exists in the Codes dialog
   - If the code type is Line or Area, an icon will display that the point belongs to a line or area. Set a string and, if required, a control code.
   - Press the **Code** button to enter information on the Point Attributes. More... You can set two control codes, and attribute values for the code.
3. Enter the **Antenna Height** and set the type of height measurement (vertical or slant). To do this, click the button and select *Edit* from the drop-down menu. The menu retains your settings. You can edit the antenna type, the value of the antenna height, and the type of height in the Antenna Setup dialog.
4. The display of the current coordinates depends on the selected coordinate system.
5. Start the measurement process according to your preference:
   - Click to perform the measurement with the precise settings and store the point either automatically or manually. The point is automatically saved, if it meets the criteria for storing the point as

specified in Settings. In manual mode, after pressing the button, two new buttons appear along with the count of the epochs achieved. Click  to save and  to cancel the measurement.

- Click  to perform the measurement with the quick settings and automatically store the point.
- Click  to open a file for data logging in surveys with post processing. More... For the user-defined file name, enter the file name at the logging start. More... Click  to stop data logging into a file.

6. If required, click  to view or change the precise and quick settings for Topo survey. More...

File or Session Name Check

This dialog requests you to select whether to Overwrite, Rename, or Just Append to an already existing file or session.

Data

The Data tab contains the information of the base and the last point that has been logged since this dialog was opened.

Map

The Map tab shows the stored points graphically. All the survey process can be done through this tab as well. It contains the information similar to the Measure tab.

Offsets

The Offsets tab allows you to determine inaccessible points by setting offsets from measured points:

Offset Line: to determine a point with the offset from a line.

Azimuth & Offsets: to determine a point with the offset from the current point.

Offset Laser: to determine a point with the offset measured by a laser device. Available if Laser has been selected in the Configuration.

Find out more on Offsets.

Line

To determine a point, which may be not physically available, relative to a reference line:

1. Define the **Reference Line** by two known or measured points:
   - Select the **Start Point** from the map or the list of the job points , or measure it by clicking .
   - Select the **End Point** of the line in the same way.
2. In the **Offsets From End Point** field enter the horizontal and vertical offsets (To define the direction for the offsets click the corresponding button.):
   - The distance from the End Point to the projection of the target point along the line.
   - The distance from the target point to the projection along the line.
   - The height difference of the target point from the End Point.
3. In the **Offset Point** field:
   - Enter the name for the calculated target point.
   - Select the code for this point from the drop-down list and, if required, click to set the point's attributes. More....
   - Click to calculate the coordinates of the new point and automatically store the point.

Azimuth Distance Height

To determine an offset point using a known or measured point (Start Point) as a reference:

1. Select the **Start Point** from the map or the list of the job points , or measure it by clicking .

2. Enter the name for the calculated target **Point**. Select the code for this point from the drop-down list and, if required, click to set the point's attributes. More...
3. Enter the angle and distance values to the target point:
   - Select between **Azimuth** and **Az to Pt** to enter the horizontal angle of the direction to the target point.
   - Select between **Zenith Angle** and **Elevation Angle** to enter the vertical angle of the direction to the target point.
   - Enter the horizontal distance between the start point and the target point.
4. Click to calculate the coordinates of the new point and automatically store the point.

Laser Measure

To determine an offset point using a laser device:

1. Select the **Occupation Point** by entering the name or select it from the map or the list of the job points . When you have the laser device with an encoder, the occupation point can be obtained from the BS dialog. More...
2. You can enter the **BS Azimuth** value, select the **BS Point**, or obtain it from the BS dialog. More...
3. In **Laser HI** enter the height of the laser device above the occupation point.
4. Enter the name of the **Point** being measured.
5. From the drop-down menu, select the **Code** for the point being measured. Click to set the point's attributes if required. More...
6. If available, click **BS Meas** to set up a new backsight if necessary. More...
7. Follow the instructions in a message which guides you on what to do and displays the measurement values.

Laser BS Meas

To set the laser device with an encoder for backsight measurements:

1. Enter the **Occupation Point** manually or select it from the map or the list of the job points .
2. Select between **BS Azimuth** and **BS Point** to enter either the backsight azimuth value or point.
3. Follow the instructions in a message which guides you on what to do and displays the measurement values.

Auto Topo

The Auto Topo dialog enables you to initiate a dynamic survey.

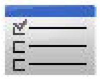
This dialog includes the following tabs:

- Auto Topo contains the initial data for the survey and conducts the survey.
- Data contains information about the stored points.
- Map allows you to perform measurements and shows the stored data graphically.

Auto Topo

The Auto Topo tab contains the initial data for the survey and displays the progress of the survey. The information about the receiver operation is shown. It duplicates the information on the Status dialog. More...

1. Enter the name of the **Point** to be logged.
2. Enter code and attribute information for the point.
   - You can select a **Code** from the drop-down list. Code needs to be defined at the time it is entered if it is not a code that exists in the Codes dialog
   - If the code type is Line or Area, an icon will display that the point belongs to a line or area. Set a string and, if required, a control code.
   - Press the **Code** button to enter information on the Point Attributes. More... You can set two control codes, and attribute values for the code.
3. Enter the **Antenna Height** and set the type of height measurement (vertical or slant). To do this, click the button and select *Edit* from the drop-down menu. The menu retains your settings. You can edit the antenna type, the value of the antenna height, and the type of height in the Antenna Setup dialog.
4. The buttons serve the following purposes:
   - to start logging points automatically with the interval specified in Settings.
   - to stop logging.
   - to interrupt the survey (active when logging).
   - to continue surveying.

-  to immediately store the current receiver position (log now).
-  to open a file for raw data logging in surveys with post processing. More... For the user-defined file name, enter the file name at the logging start. More...
-  to stop raw data logging into a file.
-  to view or change the settings for Auto Topo survey. More...

Raw Data Logging

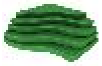
Enter the **File Name** for the file in which to log raw data.

Data

The Data tab shows the properties of the stored points: the Point IDs and their coordinates.

Map

The Map tab shows the stored points graphically. All the survey process can be done through this tab as well. It contains the information similar to the Auto Topo dialog.



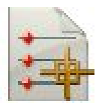
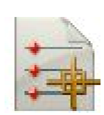
Surface

To perform survey and create or append to a real-time Surface surface:

1. Enter surface information. More...
2. Perform surface survey. More...

Volume Input

To input surface information:

1. From the drop-down button select either **New** to create a new Surface or **Append To** if you want to add the volume calculations to an existing Surface. Click  to give a name and locate the new Surface or choose an existing Surface from the list. An information message will display the maximum and minimum Northing and Easting values for the area covered by the Surface.
2. From another drop-down button select the surface to which their volumes will be computed:
   - **Boundary:** the volume calculation is performed from the Surface to a Surface created from all the points which define the boundary.
   - **Min Elevation:** the volume calculation is performed from the Surface to a flat plane created at the minimum elevation of the Surface.
   - **Max Elevation:** the volume calculation is performed from the Surface to a flat plane created at the maximum elevation of the Surface.
   - **Fixed Elevation:** the volume calculation is performed from the Surface to a flat plane created at the user entered elevation. Either manually enter the fixed elevation in the edit field or press the Map or the List selection button to select a point in the job whose elevation will be used as the fixed elevation. The edit field will display the elevation of the selected point.
   - **Plane:** the volume calculation is performed from the Surface to a flat plane created through the three points. Either manually enter the names of the three points in the edit fields or use the associated Map and List selection buttons to select these points from the job.
   - **Design Surface:** the volume calculation will be performed from the Surface to the user-entered Surface. Either manually enter the name of the second Surface or press the List selection button and choose the Surface from the list. The plots will display both of the Surface.
   - **No Volume Calc:** there will not be any volume computation performed.
3. If required, select the **Use Alignment** check box to enter a road name or select a road from a list  to

view the current station and center line offset on the map.
4. Click **Next** to perform survey. More...

Surface Topo

Perform the survey in the usual way described in regular Topo survey.

If in Volume Input you select the **Use Alnt** check box and enter a valid road, you can view the current station and center line offset, on the map and video tabs, by selecting those labels from the value selection display.

Select Volume Report from the  pop-up menu to view the stockpile/pit volume report while you are performing survey.

Map

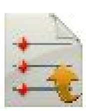
As you add points to your Surface the map view will display the Surface in the background and update it dynamically when a point is saved.

In Map properties you can clear the **Surface Surface** check box not to display the current Surface.

Surface Topo Volume Report

The volume report allows you to view the stockpile/pit volume report while you are performing survey.

- The stockpile is the volume of the Surface which sits above the plane created by the boundary of the Surface.
- The pit is the volume of the Surface which sits below the plane created by the boundary of the Surface.

If required, click  to save the volume report to a file.

Survey pop-up menu

The pop-up menu includes additional options to use in the survey, for example:

Status

Config Radio

Setup: PTL Mode, Grid Lines, **Reset RTK** - the command that reinitializes the receiver. An error may occur during the Receiver Setup.

mmGPS+ Options

Edit Points

Inverse

Show Quick Codes - when selected, shows code boxes on the Map and allows you to take measurements with these codes.

Edit Quick Codes...

Chats: Create New

Add Raw Note

Configure Radio

This dialog contains the settings for the radio modem:

- In **Radio Connected to**, select if the radio is connected to the base or the rover.
- The **Type** field tells us what the selected radio type is (it is got from the Config->Survey)

The rest of the controls on this dialog depend on the type of radio that was selected.

Beacon Status

The Beacon Status dialog displays settings for a radio beacon source for differential GPS corrections:

- The **Station** that provides broadcasting differential corrections for the rover.
- In **Status**, the Version of the Beacon Board in the receiver, the Beacon frequency and Signal/Noise ratio of the received signal are shown.

PTL Mode

The Point-To-Line mode (PTL) is a method of interpretation of the point coordinates. The coordinates are defined through the two reference points. The line trace through these points is set as one axis and its perpendicular as another.

1. In **Start Ref Point** and **End Ref Point**, enter the names of the reference points. Select these points from the map or select from the list of points .
2. Select the PTL Mode On check box to enable the PTL mode.

---

mmGPS+ Options

The mmGPS+ Options dialog allows modification of various mmGPS+ functions:

- The current status of the receiver's mmGPS+ engine is displayed as ON, OFF or DISABLED. You can change the status by using the combo box to turn the mmGPS+ engine ON or OFF.
- You can **Use weighted height computations:** The mmGPS+ height can be computed using weights for the laser and GPS data. When this box is checked, a greater weight is applied to the laser measurement when the rover is closer to the transmitter or to the GPS measurement when the rover is further away.
- In **Height Difference Limit**, set a limit for the difference between GPS and mmGPS+ height measurements. The mmGPS+ icon will change if the difference is over this limit.

Grid Setup

The Grid Setup dialog allows the setup of a grid to be displayed with the Map to help you while collecting points.

1. Select the **Display Grid** box to display a grid in the Map tab with the settings that follow.
2. Specify the **Origin Point** for the grid. Select these points from the map or select from the list of points .
3. Select either **Azimuth(Bearing)** or **Azimuth(Bearing) to Point** by clicking on the button to indicate the direction of the lines and enter the corresponding value.
4. In the **Spacing** field, specify the intervals along **y (North)** and **x (East)** axes for the grid lines.

Receiver Setup

There are shown:

- **Details** on the internal error message generated during the receiver setup.
- The **OAF Expiration On** the earliest expiring option.
- **FW Version** of the firmware in the receiver.
- In **Try Again?**, click:

  Yes to try sending the setup to the receiver again.

  No to return to the current dialog without doing anything more.

Add Raw Note

Enter a **Raw Note Text** to give a description during the survey if required.

This note is shown in Raw Data.

When creating a file from the job data, the note will be also saved in this file.

Setup Optical (Robotic) folder

Click an icon to perform the task:

Backsight

Sets up a Total Station survey with a reference direction.

Resection

Computes the coordinates of an occupation point, where the instrument is set up, using measurements to two (or more) points with known coordinates.

Benchmark

Computes the elevation of an occupation point, where the instrument is set up, using measurements to two (or more) points with known elevations.

Connect

Switches between GPS and optical instruments, and sets connections with the instrument, a network, and with MAGNET Enterprise. Not available for onboard.

Localize

Calculates localization parameters of the mathematical coordinate transformation between an original coordinate system, in which the job points are measured or given, and a local coordinate system, in which control points are known. Find out more...

Remote Ctrl

Transmits commands from the controller to the motorized total station.

Backsight

The backsight wizard helps you specify a reference direction of a Total Station survey.

1. In the **Occupation Point** group you can:
   - Enter the name of the **Point**, where the total station is located. You can set the occupied point in one of the following ways:
     - Type in the name of the point.
     - Choose the point from the map .
     - Choose the point from the list of the job points. To do this, click and select *From List* from the pop-up menu.
     - Determine an arbitrary point near an alignment. Click and select *Station and Offset*from the pop-up menu.
     - Determine the point location by resection. Click and select *Resection* from the pop-up menu.
     - Compute the elevation of an occupied point by vertical resection. Click and select *Remote BM* from the pop-up menu.
   - Type in the height of the instrument (**HI**).
   - Click to select the way to specify the scale factor for the given occupation point. You can set either Scale factor is equal one, or type in custom value to the **User Scale**.

   Note: If you have entered a new occupation point, the software prompts you to enter the point coordinates in the Add Points dialog before starting of measurement.

2. In the **Backsight Point** group you can:
   - Select between **Point** / **Azimuth** to enter either the name of the backsight location or the direction to it. You can set the backsight direction in one of the following ways:
     - Type in the name of the point.
     - Choose the point from the map
     - Choose the point from the list of the job points. To do this, click and select *From List* from the pop-up menu.
     - Determine an arbitrary point near an alignment. Click and select Station and Offsetfrom

the pop-up menu.

- Use multiple backsight points. Click and select Multiple BS from the pop-up menu.

- Enter the height of the reflector.
- Select **Fixed Height** if you want to fix the height of the backsight point for the whole set of measurements. This is useful when one target is mounted at the BS for the duration of an occupation and another is used for sideshots in mode *Ang/Dist Sets-Dir/Rev*

Note: If you have entered a new backsight point, the software prompts you to enter the point coordinates in the Add Points dialog before starting of measurement.

3. Click the **Next** button to continue.
4. Check the setup settings:
   - The **Occupied** point name and the height of the instrument.
   - The **Backsight** point name (or "---" if the **Azimuth** was selected in the **Backsight Point** group) and the height of the reflector.
   - The **Azimuth** to the backsight point location.

   The **Set Circle to** field displays the horizontal circle reading corresponding to the backsight point. The icon brings up the floating menu that enables you to set the BS Circle value to zero or azimuth, to input a value or to get from the instrument, or to change the value by +/- 90 or 180 degrees. The software retains the previous setting for this drop-down list.

   To turn a Robotic total station to the Backsight Point, click the **To BS** button.

   Select **Measure Distance** if required to measure the distance to backsight point.

   If required, click the **Check** button to take the measurement to the backsight point and then check it on the Data view.

5. Click the **Set** button to set the horizontal circle on the instrument as defined in the BS Circle field, take the measurement to the backsight point and then observe the results on the Data tab. If required, click to save the results into a file.
6. Click to observe and change the survey settings, if required. More...
7. The icon brings up the pop-up menu of additional options. More...

Notice

Click the icon provided a robotic total station is used to toggle between the status bar and the tool bar for remote control of the instrument. More...

User Scale

Enter the required value of the scale.

Pop-up menu

Additional options which can be useful are as follows:

- *View*: you can select text view (Normal) and graphic view (Map).
- *Edit Points*: opens the Points list to edit points.
- *Robotic*: opens the Remote Control Tilt dialog to control tilt correction.
- *Calc Pt to Pt*: opens the Two-Point Inverse COGO dialog.
- *Intersection*: opens the Intersection COGO dialog.
- *Raw data*: you can open either the Raw Data dialog to edit raw data or the Add Raw Note dialog to enter text of a field note.
- *Help*: opens the help topic.
- *Tilt*: opens the Remote Control Tilt dialog to control tilt correction.

Remote Control Tilt

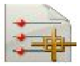
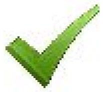
This dialog allows you to control tilt correction.

- Two bars show the level bubbles in two directions.
- X,Y fields display numerical values of the instrument tilt.
- The circle represents the tolerance of location of the level bubble for the vertical and horizontal angle tilt correction.
- If required, adjust the instrument tilt to visually set the bubble into the correction area.

Station and Offset

The Station and Offset dialog allows the backsight or occupation point to be determined by the station, offset and elevation in relation to a road.

To determine the point:

1. Click  and select the referenced alignment that can be a **Road** or **Horizontal Alignment**
2. Enter the **Station** along the road where this point should occur.
3. Enter the **Offset** from the road where this point should occur.
4. Enter the **Elevation** of the point.
5. Click  to open the Add Point dialog on which to add the calculated point to the list of points. The BS dialog opens with this occupation point.

Multiple Backsight

Multiple backsight points allow you to verify the location of the occupied point.

Take the measurements to a number of BS points on the Measurement tab:

1. Enter the first known **Point** name.
2. The **Code** field displays the point's code.
3. In **HR**, enter the height of the reflector.
4. Take measurements. More...

The Set tab displays measurements being done during one set: the Res HA (residuals of the horizontal angles) and the measured and initial parameters. Use the buttons for the following purposes:

- **Remove**: to delete the highlighted measurement from the set.
- **Re-Meas**: to replace the current measurement with a new measurement.
- **Accept**: to store the new coordinates in the database.

Taking Measurements

Depending from the selected instrument type, MAGNET Field Site offers the following ways to take measurements.

For the Conventional Total Station. When using the Conventional Total Station you may save the single measurement or an averaged value of the several measurements (Quick mode). You may also perform single or several measurements to the point; verify the result on the screen and after that save it (Precise mode).

- click  to obtain the single or the averaged measurement in Precise mode. Then click  to save it.

or

- click to take the single or the averaged measurement to the point and automatically save it.

For the Robotic Total Station. The Robotic Total Station automatically takes the measurement to the target. You can save the last measurement (Quick mode), or perform single or several measurements to the point; verify the result on the screen and after that save it (Precise mode).

- click to obtain the single or the averaged measurement in Precise mode.

or

- click to save the last measurement in Quick mode or to save the averaged measurement in Precise mode.

For the Robotic Total Station onboard:

- click to start measurement and to obtain the single or the averaged measurement in Precise mode.

or

- click to take the single measurement to the point and automatically save it.

Resection

The method of resection allows you to determine the location of the Total Station occupation point by measurement of the known points.

The resection wizard helps you perform the resection:

1. In the **Define instrument setup** group you can:

   - Set a name of the occupation point **(Occupy**). If you determinate the coordinates of an unknown occupation point, type in the point name. If you want to recalculate the coordinate of the known occupation point choose the point either from the map or from the list
   - Type in the height of the instrument (**HI**)
   - Select from the list (**Code**) a desire code for the occupation point.
2. Click the **Next** button to continue.
3. In the **Specify another control point** group you can:

   - Enter the known name (**Point**); you can select the point either from the map or from the list .
   - Type in the height of the reflector (**HR**)
   - The **Measure FS Direct** displays the values of horizontal angle (HA), vertical angle (VA) and slope distance (SD).
   - Take the measurements. More...
4. Repeat the procedure for the remaining known points.
5. After saving second point (and subsequent point) the Results screen displays measured value of angles and slope distances with estimate of accuracy. In this screen you can:

   - Click **Add** to continue measure an other control point.
   - Click **Remove** to delete any selected measurement.
   - Click **Re-Meas** to repeat the measurement to the previous point.
   - Click **Accept** to calculate (or recalculate, when you add a measurement to next control points) the occupation point coordinates and finish the resection procedure.
6. After finishing the resection procedure you can add a new measurement for the given occupation point. Click and perform a new TS measurement.

By using Resection Options you can select resection either in the horizontal plane only (2D) or the horizontal and vertical plane (Resection 3D).

If you activated Hybrid Positioning mode, the Resection for this mode will prompt to perform TS and GPS measurement for the point.

Resection 3D

To calculate the default three dimensional resection, minimum two points with distance measurements are required. With angle-only measurements, three points are required. If additional points are measured, a least-squared solution is used. To change the type of resection to use only horizontal coordinates (2D), click  and select Resection Options from the pop-up menu. The 2D/3D option is retained between sessions. When doing a resection next time, the resection will start up with the previous used setting.

To perform the resection, see the steps in the Resection. The Result screen displays the result of the sideshots being done.

The icon  brings up the pop-up menu of additional options. More...

Resection in Hybrid Positioning mode

The Hybrid positioning mode allows you to simultaneously record the point coordinates from GPS rover receiver and Total Station to the opened job. This option can be activated only for Robotic Total Station with reflector. The GPS receiver and Robotic Total Station are connected to the job. Using Hybrid Positioning mode you can obtain an occupation point coordinates in Grid or WGS-84 coordinate systems.

Before performing resection you need activate **Hybrid Positioning**, select desired prism and select **Hybrid positioning antenna offset** check box and open **Resection** dialog.

There are four different scenarios to perform the resection procedure:

A. The ***desired Grid projection is defined in the job***. ***Grid to Ground transformation is not selected***. The GPS and TS measurements are performed for each ***unknown*** point. The coordinates of the Occupation Point are calculated in the current Grid coordinate system.
   1. Select the desired Grid projection in the Coordinate System dialog.
   2. In the **Define instrument setup** group (**Resection** dialog) you can:

- Set a name of the occupation point **(Occupy**). If you determinate the coordinates of an unknown occupation point, type in the point name. If you want to recalculate the coordinate of the known occupation point choose the point either from the map or from the list .
- Type in the height of the instrument (**HI**)
- Select from the list (**Code**) a desire code for the occupation point.

3. Click the **Next** button to continue.
4. In the **Specify another control point** group you can:

- Enter the name (**Point**); you can select the point either from the map or from the list .
- Type in the height of the reflector (**HR**)
- The **Measure FS Direct** displays the values of horizontal angle (HA), vertical angle (VA) and slope distance (SD).
- Take the TS measurements. More...

5. In the **Measuring GPS for ...** you see the point coordinates in the current Grid are measured by the GPS receiver. In this group you can:

- Change the point name or select the point either from the map or from the list .
- Click to perform the measurement with the precise settings and store the point either automatically or manually.

or

- Click to perform the measurement with the quick settings and to automatically store the point.

6. After saving second unknown point (and subsequent point) the Results screen displays measured value of angles and slope distances with estimate of accuracy. In this screen you can:

- Click **Add** to continue measure an other control point.
- Click **Remove** to delete any selected measurement.
- Click **Re-Meas** to repeat the measurement to the previous point.
- Click **Accept** to calculate (or recalculate, when you add a measurement to next control points) the occupation point coordinates in the current Grid coordinate system and finish the resection procedure. Clicking on the button opens the **Store Point** dialog, where the occupation point coordinates are displayed.

7. After finishing the resection procedure you can add a new measurement for the given occupation point. Click and perform a new TS / GPS measurement.

B. The ***desired Grid projection is defined*** in the job. ***Grid to Ground transformation is selected***. The relation ship between Grid and Ground coordinate systems is known. Coordinates for both coordinate sets are displayed in the current coordinate system. The coordinates of the Occupation Point are calculated in the Grid or Ground coordinate systems. The routine of the resection procedure is equal previous scenario.

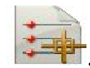
1. Select the desired Grid projection in the Coordinate System dialog.
2. Click the Use Grid/Ground check box, click and select the desired parameter (**Parameters**) in the Grid/Ground Parameters dialog. Click to save the selected grid to ground parameters.
3. Perform the steps 2-6 in the previous scenario.

C. The ***Grid projection is not defined*** in the job. The user has coordinates of the control points (two points at least) in the Ground coordinate system. For each control point TS and GPS measurements are provided. The software automatically performs the localization between WGS-84 and Ground. The coordinates of the Occupation Point are calculated in the Ground and WGS-84 coordinate systems.

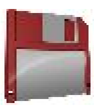
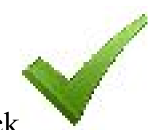
1. Select <none > in the **Projection** of the Coordinate System dialog.
2. In the **Define instrument setup** group (**Resection** dialog) you can:
   - Set a name of the occupation point **(Occupy)**. If you determinate the coordinates of an unknown occupation point, type in the point name. If you want to recalculate the coordinate of the known occupation point choose the point either from the map or from the list .
   - Type in the height of the instrument (**HI**)
   - Select from the list (**Code**) a desire code for the occupation point.
3. Click the **Next** button to continue.
4. In the **Specify another control point** group you can:
   - Either enter the name of the new point (**Point**) or select the known point from the map or from the list .
   - Type in the height of the reflector (**HR**)

- The **Measure FS Direct** displays the values of horizontal angle (HA), vertical angle (VA) and slope distance (SD).
- Take the TS measurements. More...

Note: If you have entered a new point, the software prompts you to enter the point coordinates in the Add Points dialog before starting of measurement.

5. In the **Measuring GPS for ...** you see the point coordinates in WGS-84 are measured by the GPS receiver for localization. In this group you can:
   - Change the point name or select the point either from the map  or from the list .
   - Click  to perform the measurement with the precise settings and store the point either automatically or manually.

     or

   - Click  to perform the measurement with the quick settings and to automatically store the point.
6. After saving second known point (and subsequent point) the Results screen displays measured value of angles and slope distances with estimate of accuracy. In this screen you can:
   - Click **Add** to continue measure an other control point.
   - Click **Remove** to delete any selected measurement.
   - Click **Re-Meas** to repeat the measurement to the previous point.
   - Click **Accept** to calculate (or recalculate, when you add a measurement to next control points) the occupation point coordinates in the current Ground coordinate system and finish the resection procedure. Clicking on the button opens the **Store Point** dialog, where the occupation point coordinates are displayed. Click  to perform the localization between Ground and WGS-84 coordinate system. If the factual residuals for the known point less than internal threshold values, the localization will be created. After that:
     - In the Projection field Localization is set automatically.
     - Next measured pair of points is not automatically added to localization. The localization parameters are not updated.
     - Next measured pair of points is used in recalculation of the occupation point coordinates.

Note: When the Localization is selected as the current projection of the job, you can use unknown point(s) for resection task in the Hybrid Positioning mode.

You can see the pairs point are used for localization in the Localization Result dialog and you can manually add or delete any pairs point.

7. After finishing the resection procedure you can add a new measurement to known point. Click  and perform a new TS / GPS measurement. The point coordinates in the Ground and WGS-84 coordinate systems will be added to localization automatically.

D. The coordinates of the control points (two points at least) in ***the Ground coordinate system*** imported in the job. The desired ***Grid projection is defined*** in the job. For each control point TS and GPS measurements are provided. The software automatically performs the localization between Grid and Ground. The coordinates of the Occupation Point are calculated in the Grid and Ground coordinate systems.

1. Import the set of points with coordinates in Ground coordinate system to the job. Then select the desired Grid projection.
2. Perform the steps 2-6 in the previous scenario.
3. After finishing the resection procedure you can add a new measurement to known point. Click  and perform a new TS / GPS measurement. The point coordinates in the Ground and Grid coordinate systems will be added to localization automatically.

Results Screen

The tab contains the data collected during the measurements, grouped by sets (one set for Multiple mode contains two measurements; one set of the Repeat mode contains four measurements).

The columns are:

- **Point**: the name of the point
- **Res HA**: Difference of each HA measurement within the set from the average of all the HA's in the set
- **Res VA**: Difference of each VA measurement within the set from the average of all the VA's in the set
- **Res SD**: Difference of each SD measurement within the set from the average of all the SD's in the set
- **H**, **V**, **SD**: Check marks indicate that the Horizontal angle, Vertical angle and the Slope Distance were used in resection. Use the Use Ctrl button to toggle.
- **HR**: The height of reflector
- **HA**: Horizontal Angle measurement within the corresponding set
- **VA**: Vertical Angle measurement within the corresponding set

- **SD**: Slope Distance measurement within the corresponding set
- **Set**: The number and type of measurement within the corresponding set

If there is sufficient data to compute the resection, the results of resection are displayed: standard deviations for coordinates and calculated scale factor.

The buttons serve the following purposes:

- **Accept**: to save the new point.
- **Re-Meas**: to replace the current measurement with a new measurement.
- **Remove**: to delete the selected measurement.
- **Use Ctrl**: to toggle through specific measurements in the resection, for example the horizontal angle, but not the vertical, or vice versa.

Pop-up menu

Additional options which can be useful are as follows:

- *Setup: PTL Mode, Grid Lines*
- *Edit Points*: opens the Points list to edit points.
- *Calc Pt to Pt*: opens the Two-Point Inverse COGO dialog.
- *Options*: opens the Resection Options dialog.
- *Add Raw Note*: opens the Add Raw Note dialog to enter text of a field note.
- *Show Quick Codes*: when selected, shows codes on the Map and allows you to take measurements with these codes in Quick mode in Topo survey and to log now in Auto Topo survey by clicking the code.
- *Edit Quick Codes*: opens the Code Options dialog to define quick codes.

Resection Options

The Resection Options dialog lets you control the input parameters and options for the implementation of the least squares solution.

1. From **Resection Type**, select *2D* to use only horizontal coordinates or *3D* to use heights as well.
2. From **Resection Method**, for 3D resection, select *2D+H* to split the least squares solution into horizontal and vertical solutions or *3D Combined* to perform it as a single 3D solution.
3. If required, clear the **Use Default Measurement Accuracy** check box to change the system default values for measurement accuracy:

- In **Distance**, error in the distance measurement.
- In **PPM**, part per million error in the distance measurement.

- In **Horz Angle**, error in the horizontal angle measurement.
- In **Vert Angle**, error in the vertical angle measurement.

Benchmark

The method of elevation allows you to determine the vertical location of an occupied point by resection from two (or more) known vertical locations.

Set the occupied point:

1. In **Occupy**, enter the name of the point for which the elevation will be computed. You can set the point in one of the following ways:
   - Type in the name of the point.
   - Choose the point from the map .
   - Choose the point from the list of the job points. To do this, click and select *From List* from the pop-up menu.
   - Determine an arbitrary point near an alignment. Click and select *Station and Offset*from the pop-up menu.
2. Enter the height of the instrument (**HI**) and the height of the reflector in the current units.
3. Click **Next** to proceed on the Known Elevation dialog.

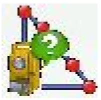
Known Elevation (Multiple)

Take the measurements to known vertical locations on the **Measurement** tab:

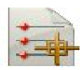
1. Enter the known vertical location in one of the following ways: click the button to choose either **Point** or **Elevation**.
   - You can select the known **Point** either from the map or from the list . The **Code** field will display the code information on the selected point.
   - Enter the value of the known **Elevation**.
2. Enter the height of the target. The field saves the entry from the Elevation dialog.
3. Take measurements. More...

The **Set** tab displays the result of the sideshots being done during one set. Use the buttons for the following purposes:

- **Remove**: to delete the highlighted measurement from the set.
- **Re-Meas**: to replace the current measurement with a new measurement.
- **Accept**: to store the new coordinates in the database.

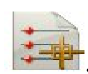
 Reference Line Setup

Set the occupation point:

1. In **Point**, enter the name of the point where the instrument is set up. It can be either a new occupation name or a known point to refresh. You can enter the point in one of the following ways:
   - Type in the name of the point.
   - Choose the point from the map .
   - Choose the point from the list  of the job points.
2. Enter the height of the instrument (**HI**) in the current units.
3. Click **Next** to select a first design point. More...

Reference Line Point

Select a design point for setting the reference line:

1. Select the **Point** by typing its name, selecting it from the map  or from the list .
2. Optionally, enter the offset values from the point in horizontal and vertical planes:
   - As required, set either **Right** or **Left** offset relative to the occupation point in the line direction.
   - Set the value of the vertical offset: select the type from **Up/Down/Elevation**.
3. The plot below will schematically show your selections.
4. Click **Next** to take measurements. More...

Point Measurements

Setup the target point and sight to it:

1. Enter the height of the target **HR**.
2. Click  to take measurements. More...

3. Observe the results of calculation. More...

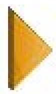
Reference Line Results

The dialog shows the setup information graphically. The instrument location is solved based on measurements, the azimuth between the two points, and the hold location.

1. Observe the reference error.
2. Select which point should be held:
    - **Hold Right** to hold the first point
    - **Hold Left** to hold the second point
    - **Hold Center** to hold the midpoint
3. Click **Accept** to store the new coordinates in the database.
4. Click **Back** to return to the previous dialog.

 Remote Control

From the Remote Control dialog, you can control the total station through the radio. It shows the current values of the total station measurements and provides a set of tools for the control. If the instrument chosen is robotic, all the observation dialogs have such tools to provide remote control.

Click the icon  to toggle between the status bar and the tool bar.

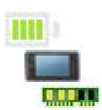
The status bar includes the following icons:

 Shows the power level of the instrument.

 Shows the status of connection with the instrument.

 Shows the prism constant for measurements with the reflector.

 Indicates reflectorless measurements.

 Shows the power and memory level in the controller.

The tool bar includes icons of control commands. Click the icon to execute a required command:

Qlock
Cause the Total Station to search for an RC device. RC is the Remote Control System for optical communications. For instructions on how to operate the RC devices, consult the instruction manual for the corresponding RC.

Turn
Opens the Rotate dialog which allows the Total station to turn to various angles or points.

Control arrows
Allows turning the total station on devices without arrow keys. Each button corresponds to a direction to turn, the button in the center can be used to stop turning.

Search

Makes the instrument search for the prism.

Lock

Locks onto the prism or tracks it.

Stop

Makes the total station stop tracking the prism and go into a "Standby" mode. The Data Indicator above the Qlock button shows the current state of the Total Station. There are four types: no data, querying status, turning, and receiving data.

The Data Indicator shows the current state of the Total Station. There are four types: no data, querying status, turning, and receiving data.

Rotate

You can rotate the remote total station by angles or to a point:

- By **Rotation Angles**:

1. The **Hz** and **Vert** fields display the current angles. Enter the values of the horizontal and vertical angles to turn. Optionally you can click to add or subtract 90 or 180 degrees to/from shown values.
2. Click **Turn** to send the data to the total station. The corresponding icon shows the rotation process.

- **Rotate to Point**

1. Select a point by typing its name, selecting it from the map or from the list .
2. Enter the HR value (the height of the reflector).
3. Click the **Turn** button.

- Click **Plunge TS** to perform "Plunge" or "Flip" (rotate the telescope and the body by 180 degrees).
- The icon lets you to turn the total station on devices without arrow keys. Each button corresponds to a direction to turn, the button in the center can be used to stop turning.

Map

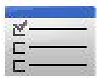
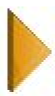
The Map tab shows the points in the graphic mode. The buttons on the right duplicate the controls on the first tab.

Topo Survey

The optical topo survey determines sideshot points in one of three modes:

- Sideshot-Direct
- Sideshot-Direct/Reverse

On any survey dialog you can:

- Click  to change the sideshot mode, if required. More...
- Click the **EDM** button to select the distance measurement mode as required.
- Click the icon  provided a robotic total station is used to toggle between the status bar and the tool bar for remote control of the instrument. More...
- Click the icon  to open the pop-up menu of additional options. More...

Pop-up menu

Additional options depend on the instrument used. They can include:

- *Setup:* Backsight, PTL Mode, Display Coords (select to display coordinates of the point when measured), Grid Lines
- *Adv*: If one point is tagged as Traverse Point, opens the Backsight dialog that automatically set the traverse point as the next occupation point. The current occupation point becomes the next backsight point. If there is more traverse points, a list of these points displays. More...
- *Edit Points*: opens the Points list to edit points.
- *Inverse*: opens the Two-Point Inverse COGO dialog.
- *Robotic*: Remote Settings, Tilt.
- *Image*: Mirror Image, Record Wide Image, Record Tele Image (telescope view with the crosshair), Image as SXGA (standard of resolution).
- *Add Raw Note*: opens the Add Raw Note dialog to enter text of a field note.
- *Show Quick Codes*: when selected, shows code boxes on the Map and allows you to take measurements with these codes in Quick mode in Topo survey and to log now in Auto Topo survey by clicking the code.
- *Edit Quick Codes*: opens the Code Options dialog to define quick codes.

Advance

This dialog allows you to automatically update the Backsight dialog for the next occupation point if more than one point have been tagged as Traverse Points.

The *Advance* option displays a list with all tagged Traverse points to **Select Next Occupation Point**. The current occupation point will be the backsight point.

Sideshot-Direct

The Sideshot-Direct dialog contains the initial data for single sideshots and displays the information during survey. The measurement to a single point is taken using the Direct position of the Total Station.

To perform surveying:

1. Enter the name of the current **Point**. During the survey the numerical part of the name changes automatically by one.
2. Set **Code** and attribute information for the point. Code needs to be defined at the time it is entered if it is not a code that exists in the Codes dialog
   - You can enter a **Code** manually or select from the drop-down list.
   - If the code type is Line or Area, an icon will display that the point belongs to a line. Set a string and, if required, a control code.
   - Press the **Code** button to view information on the Point Attributes. More... You can set two control codes, and attribute values for the code.
3. Enter the height of the reflector (**HR**).
4. Select the **Traverse Point** check box to tag the measured points as traverse.
5. Take measurements. More...
6. Observe the results on the Data tab and on the Map. The Map lets you also perform measurements.
7. The Offsets tab allows you to determine inaccessible points by setting offsets from measured points. More...

Sideshot-Direct/Reverse

The measurement to a single point is taken using the Direct Position and the Reverse Position of the Total Station (i.e., Plunge - Flip and Rotate the Total station by 180 degrees to get the reverse measurement). One set consists of one direct and one reverse measurement. These measurements are used to eliminate the Vertical circle centering errors.

This dialog behaves similar to the Sideshot-Direct dialog except that one measurement constitutes a set of readings rather than a single one.

Offsets

Switch to an appropriate offset task as required:

Hz Angle
Determines a point using the horizontal angle from one point and the distance to another.

Hz-Vt Angle
Determines a point using the horizontal and vertical angles.

Distance Offset
Determines a point giving you the ability to add or subtract distances, horizontally and vertically.

Hidden Point
Determines a point on the ground surface with a slanted rod touching the ground point.

2 Line ISection
Determines a point by the intersection of the two lines. Each line is defined by two points or two measurements.

Line & Corner
Determines a point on the corner using one line defined by two points and a horizontal angle measurement.

Line & Offset
Determines a point that is in a distance from a line defined by two points.

Plane & Corner
Determines a point (Corner) by a plane defined by three points and horizontal and vertical angle measurements.

Find out more on Offsets.

Hz Angle Offset

The Measurement tab of the Horizontal Angle Offset dialog contains data for definition of a point using the horizontal angle from one point and the distance to another.

1. Enter the name for the offset **Point** to be stored.
2. Enter a **Code** for the offset point. Can be entered manually or chosen from the drop-down list.
3. Set **HR** above the mark (rod height).
4. Take measurements in any order:

- You should measure to Center and obtain the vertical angle and horizontal angle measurements.
- A Side measurement provides VA, HA, and distance measurements. With these two sets of measurements, the computation can be made for point at center of a tree.

Find out more on Offsets.

Hz/Vt Angle Offset

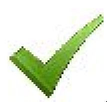
The Survey Offsets dialog in the Hz-Vt Angle mode contains data for definition of a point using the horizontal and vertical angles and a distance.

1. Enter the name for the offset **Point** to be stored.
2. Enter a **Code** for the offset point. Can be entered manually or chosen from the drop-down list.
3. Set **HR** above the mark (rod height).
4. Measure the slope distance to target.
5. Combine the horizontal angle and vertical angle measurements with distance logged in Distance step to determine the point location.

Find out more on Offsets.

Distance Offset

The Distance Offset dialog contains data for the definition of a point giving the ability to add or subtract distances, horizontally and vertically.

1. Enter the name for the offset **Point** to be stored.
2. Enter a **Code** for the offset point. Can be entered manually or chosen from the drop-down list.
3. Set **HR** above the mark (rod height).
4. Take the measurements.
5. Enter Distance Offsets in the next dialog and click  .

Find out more on Offsets.

Enter Distance Offsets

Enter the Distance Offsets in this dialog after taking the measurements.

1. **Away/Toward** - the distance between the current point and the projection of the offset point on the line of sight.

2. **Right/Left** - the distance between the offset point and its projection, taking into consideration its location in regard to the line of sight. If required, select the **Right/Left Offset: Rod to TS** check box to change the direction.
3. **Up/Down** - the height of the point in regard to the current position.

Hidden Point

The Measurement tab of the Hidden Point dialog allows you to define a point on the ground surface with a slanted rod touching the ground point. The rod has two targets.

1. Enter the name for the offset **Point** to be stored.
2. Enter a **Code** for the offset point. Can be entered manually or chosen from the drop-down list.
3. Set **HR** above the mark (rod height).
4. Take measurements in any order:

- Measure the first target on the rod.
- Measure the second target on the rod.

Find out more on Offsets.

Two Lines Intersection

The Intersection dialog contains data for determination of a point by intersection of two lines. Each line is defined by two points or two measurements.

1. Enter the name for the offset **Point** to be stored.
2. Enter a **Code** for the offset point. Can be entered manually or chosen from the drop-down list.
3. Set **HR** above the mark (rod height).
4. Take measurements in any order:

- Measure the first point defining the first line.
- Measure the second point defining the first line.
- Measure the first point defining the second line.
- Measure the second point defining the second line.

Find out more on Offsets.

Line and Corner

The Line and Corner dialog contains data for determination of a point on the corner using one line defined by two points.

1. Enter the name for the offset **Point** to be stored.
2. Enter a **Code** for the offset point. Can be entered manually or chosen from the drop-down list.
3. Set **HR** above the mark (rod height).
4. Measure Line Points:

- Measure the first point defining a line.
- Measure the second point defining a line.
- Measure the horizontal angle to locate a point on the line at the corner.

Find out more on Offsets.

Line and Offset

The Line and Offset dialog contains data for the determination of a point distant from a line defined by two points.

1. Enter the name for the offset **Point** to be stored.
2. Enter a **Code** for the offset point. Can be entered manually or chosen from the drop-down list.
3. Set **HR** above the mark (rod height).
4. Take measurements in any order:
    - Measure the first point on the line.
    - Measure the second point on the line.
5. Enter **Distance Offsets** and click  :
    - *Away/Toward* - the distance between the current point and the projection of the offset point on the line of sight.
    - *Right/Left* - the distance between the offset point and its projection, taking into consideration its location relative to the line of sight.
    - *Up/Down* - the height of the point relatively to the current position.

Find out more on Offsets.

Plane and Corner

The Plane and Corner dialog helps you to determine a point (Corner) by a plane defined by three points and an angle measurement.

1. Enter the name for the offset **Point** to be stored.
2. Enter a **Code** for the offset point. Can be entered manually or chosen from the drop-down list.
3. Set **HR** above the mark (rod height).

4. Measure **Plane Points**. Note that three points defining the plane must not be collinear (all on the same line):
    - Measure the first point in a plane.
    - Measure the second point in a plane.
    - Measure the third point in a plane.
5. Obtain horizontal and vertical angle measurements to determine the point in a plane at the corner.

Find out more on Offsets.

Surface

To perform survey and create or append to a real-time Surface surface:

1. Enter surface information. More...
2. Perform surface survey. More...

   If in Volume Input you select the **Use Alnt** check box and enter a valid road, you can view the current station and center line offset, on the map and video tabs, by selecting those labels from the value selection display.

   Select Volume Report from the pop-up menu to view the stockpile/pit volume report while you are performing survey.

Auto Topo

This function is activated only with Robotic instruments and allows you to collect points by Time or Distance according to your preference to conduct the survey.

1. The **Point** field displays the current point name. You can change it.
2. From the drop-down menu, select the **Code** for the point being measured. Click to set the point's attributes if required. More...
3. Enter the **HR** (the height of the reflector).
4. Click to start the survey process. After pressing, the button changes to to stop the process.
5. Click to immediately store the current position.

For a description of other buttons, refer to Remote Control.

 Stake Points

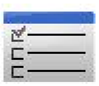
To stake a point:

1. Enter the name of the **Design Point** to be staked. Enter the point manually or select it from the map or from the list of the job points.
2. The **Code** field displays the code for the design point.
3. For GPS, the **Antenna Height** shows the default height of the antenna reference point (ARP) above the mark. You can edit the antenna type, the value of the antenna height, and the type of height in the Antenna Setup dialog. To do this, click the **Ant Ht** button, and select *Edit*.
4. For TS, in **HR**, enter the reflector height.
5. **Stake Report** shows the name of the current stake report if it is set.
6. Click **Stake** to open the Stake dialog, assisting the stakeout process. More...

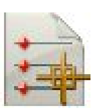
For the icons available on the dialog see Icons Description

Icons Description

Brings up the pop-up menu of additional options. More....



Opens the stake settings to view/edit. More....



Opens the list of points to choose the point.

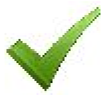
Opens a map to choose the point.

Indicates the vertical type of the antenna height.

Indicates the slant type of the antenna height.

Sets the height of the reflector.

Opens the report list to view/change the report. More....



Confirms settings, closes the dialog, and returns to the previous dialog.

Stake pop-up menu

The pop-up menu includes additional options which can be useful during the stakeout. The items with small arrows have submenus. The menu content depends on the instrument used:

- Stake pop-up menu for GPS.
- Stake pop-up menu for Conventional Total Station.
- Stake pop-up menu for Robotic Total Station.

Stake pop-up menu for GPS

Clicking on such a menu item opens a subset of items.

1. **Setup** - click to expand the menu:
   - Status - opens the dialog which contains information about the current position of the receiver, RTK status, and the satellite constellation.
   - Config Radio - opens the dialog which contains the settings for the radio modem.
   - PTL Mode - if this mode is selected, the list and map will display only the job points having PTL coordinates for selection. For this mode the design point for stakeout is a PTL (point to line) point.
   - **Reset RTK** - the command that reinitializes the receiver and reload the current survey configuration. An error (Receiver Setup) may occur during the receiver reinitialization.
2. **Edit Points** - opens the dialog which displays the list of the job points in the current coordinate system.
3. **Calc Pt to Pt** - opens Two-Point Inverse dialog where you can calculate the azimuth and distance between two known points.
4. **View Report**-opens the current report.
5. **Raw Data**- click to expand the menu:
   - **Add Raw Note** - opens the Add Raw Note dialog where you can enter any description during the survey if required.
   - **Edit** - opens the Raw data dialog with raw data collected in the job.
6. **Chats**

- **Create New** - opens the Chat dialog is displayed. In this window you can select a user or a group of users of your company and create a new chat.

7. **Help** - opens the help topic about staking points.

Stake pop-up menu for Conventional Station

Clicking on such a menu item opens a subset of items.

1. **Setup** - click to expand the menu:
   - Backsight - you can set the BS direction angle.
   - PTL mode - if this mode is selected, the list and map will display only the job points having PTL coordinates for selection. For this mode the design point for stakeout is a PTL (point to line) point.
2. **Edit Points** - opens the dialog which displays the list of the job points in the current coordinate system.
3. **Inverse** - opens Two-Point Inverse dialog where you can calculate the azimuth and distance between two known points.
4. **View Report**-opens the current report.
5. **Raw Data**- click to expand the menu:
   - **Add Raw Note** - opens the Add Raw Note dialog where you can enter any description during the survey if required.
   - **Edit** - opens the Raw data dialog with raw data collected in the job.
6. **Chats**
   - **Create New** - opens the Chat dialog is displayed. In this window you can select a user or a group of users of your company and create a new chat.
7. **Help** - opens the help topic about staking points.

Stake pop-up menu for Robotic Total Station

Clicking on such a menu item opens a subset of items.

1. **Setup** - click to expand the menu:
   - Backsight - you can set the BS direction angle.
   - PTL mode - if this mode is selected, the list and map will display only the job points having PTL coordinates for selection. For this mode the design point for stakeout is a PTL (point to line) point.
   - Drape mode - when need to stakeout a point, which is located inside a wall, tunnel, or road, we recommend to activate Drape Mode for the robotic reflectorless total station (**Target Type** – *Non Prism*).
2. **Robotic** - click to expand the menu:
   - Config Link - opens Remote Connection dialog to view or modify configuration of remote connection of the instrument with the RC device and the data controller.

- Tilt - opens the dialog with two bars show the level bubbles in two directions.

3. **Edit Points** - opens the dialog which displays the list of the job points in the current coordinate system.
4. **Inverse** - opens Two-Point Inverse dialog where you can calculate the azimuth and distance between two known points.
5. **View Report**-opens the current report.
6. **Raw Data**- click to expand the menu:
   - **Add Raw Note** - opens the Add Raw Note dialog where you can enter any description during the survey if required.
   - **Edit** - opens the Raw data dialog with raw data collected in the job.
7. **Chats**
   - **Create New** - opens the Chat dialog is displayed. In this window you can select a user or a group of users of your company and create a new chat.

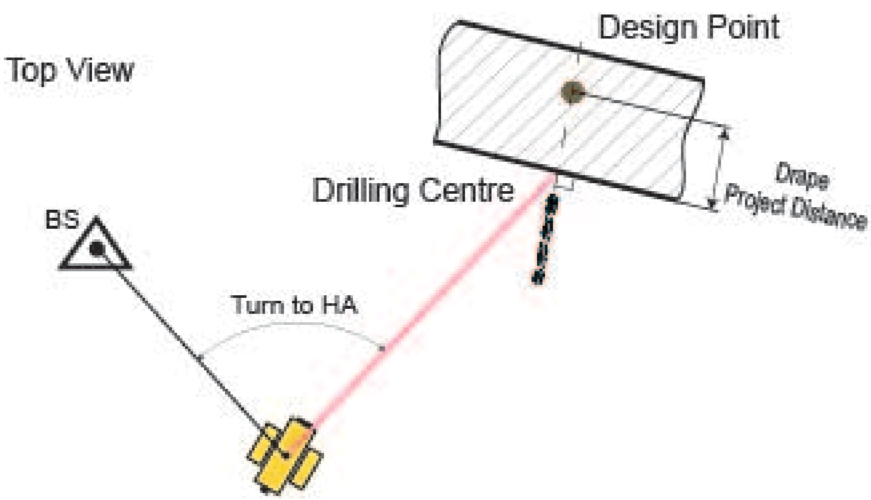
Drape Mode

After activating Drape Mode, the total station will automatically scan a surface near the hidden point. The result of the scanning is the defining of the 3D surface orientation near this point. By using this data, MAGNET Field Site calculates:

- Coordinates of the center of drilling on the surface. The laser pointer will show the center on the surface.

- The value of the depth drilling from the center to the hidden point.

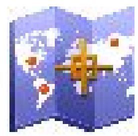
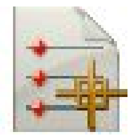
Note: A drill bit is set perpendicular to the surface.



See How to stake out a hidden point.

How to stake out a hidden point

To stakeout the hidden (design) point, take the following steps:

1. Select BS point (**Setup** – **Backsight**).
2. Click  in the main menu.
3. Click  in the stake menu.
4. Click Help Options icon (), select Setup and select Drape Mode from the list.
5. Select the desired design point from the map (click ) or from the list (click ) and click the Stake button. After clicking tapping the button:
   - Robotic total station will automatically turn to the hidden point direction and will scan a surface near the hidden point. The scanning procedure will take several minutes.
   - The Stake screen will be opened (by default the Stake Map is opened).
   - For the activated Drape Mode the Drape Project Distance line will be added to the list of the **Select Value** dialog. Check this line to display the value on the Stake screen.
   - The button with design point name will appear at the bottom of Stake screen.
6. After finishing scanning procedure, the coordinates of drilling center and value of the drilling depth are calculated:
   - The button with design point name is renamed to "design point_drp". To obtain information about point coordinates, click tap this button.
   - The Laser Pointer will show the drilling center on the surface.
   - The Drape Project Distance field displays the value of the depth drilling from the center to the design point.

Stake

The Stake dialog is designed to assist in the stakeout process. The stake dialog contains a number of flexible features which allow you to tailor and customize your staking requirements.

The main dialog consists of:

- Instrument panel along the top,
- Stake panel along the bottom and

- View. Depending on the current staking application there are a number of different view options which provide different graphical aids for staking.
- With the exception of the data view there are also four selectable Data fields to display specific stakeout information.

Instrument Panel

The Instrument panel along the top of the dialog displays GPS/Optical instrument specific iconic information. This information can be displayed or hidden by pressing the arrow button on the left of the dialog.

Stake Panel

The Stake Panel along the bottom of the dialog displays a number of buttons used to provide information and facilitate staking.

- **Design Button:** The design button on the left provides the information on the current design point. Pressing the button will display a pop-up list detailing the design point information.
- **Arrow Buttons:** The arrow buttons are displayed if the staking application permits multiple point locations. Pressing the buttons will advance forward and backward through the different design locations.
- **Reading Button** (for TS)**:** The read button is used to make an observation.
- **Measuring Button** (for GPS):The measure button is used to take a measurement.
- **Store Button:** The storage button located to the far right is used to save the current staked observation.

Data Fields

All views except the Data view have four selectable data fields used to display specific staking information. To change one of these data fields simply press on the field and select from a list of all the available fields for the current stake module. The Data view always contains all the available data fields.

Views

Each of the staking modules has a number of different views to provide different aspects to help the staking process. To access the different views either press and hold on the stake dialog until the view popup appears for a new selection or click View Panel on the pop-up menu .

The following views can be available:

- Data View
- Map View
- Normal View
- Overhead View
- Cross Section View
- Surface View

Data View

The Data view is accessible on all stakeout modules. The Data view provides a detailed list of all data fields available for the current staking module.

Map View

The Map view is also accessible on all stakeout modules. When staking in the map view the background image of the stakeout is the job's map. A map menu panel is available which provides the map controls, such as zoom.

Normal View

The Normal view is available whenever you are staking known design coordinates. This view provides a directional way of locating the design locations. If the distance to the design is more than 3 m, a blue arrow will point to the target direction with the current location in the center of the dialog. If the distance to the target is less than 3 m, the graphic shows the target point in the center and the current location. As soon as the target becomes closer than the Horizontal Distance Tolerance value the graphic shows a bull's-eye target point on the dialog.

Overhead View

The Overhead view is also available whenever you are staking known design coordinates. This view is similar to a map view and shows the current location and design location.

Cross Section View

The Cross Section view is available when staking roads. This view shows the current stake location in a vertical display as well as the road cross section. A map menu panel is available which provides the map controls, such as zoom.

Surface View

The Surface view is available when staking Surface. This view shows the surface and current stake location. A map menu panel is available which provides the map controls, such as zoom.

Design Offsets

The Height dialog is available in the Stakeout Point in the upper-left menu by selecting **Set Ellipsoidal ht**, **Set Elevation**, or **Set Z** depending on the coordinate system and display type. The dialog enables you to change the height of the current point.

- In **Design Height/Elevation** select the check box to apply the Design Height/Elevation for the point to be staked and enter the elevation to be used.
- In **Road Offset** select the check box to apply the Road Offset to the height of the point to be staked and enter the offset value to be used.
- In **Surface Offset** select the check box to apply the Surface Offset for the Surface selected to be staked and enter the offset value to be used.

Set References / Surface and Point References

This dialog is accessible through the Stakeout Views and the Store Point dialog when the type of staking uses cross sections (for example roads). The dialog displays the current cross section, graphically displayed with all points shown with check marks, the corresponding label to each point, and lines connecting the points.
Each point and line is selectable and will be marked with a green color if selected.

- Click  to store the references.
- All selections can be cleared. To do this click  and select **Remove selected**.

Design Pt/Layer

- Select the **Display Store Pt Info** check box to display the staked point before storing it.
- In **Layer**, select the layer to which this point is to belong. If required, click  to edit the layer.

Initial Point Name

The Initial Point Name dialog enables you to specify the starting name for the points calculated for the stakeout task.

1. In **First Name**, enter the name of the first point.
2. Select the **Code** and attributes of the points. The code can be chosen from the list or entered manually.
3. In **Point Inc/Dec Size**, specify the increment or decrement of the name of the next design point.

Stake Surface

Stake Surface enables you to stake a surface as a known fixed elevation.

Select one of three types of Stake Surface as required: Elevation, Road, or Surface.

Stake Surface Elevation

To stake a surface of a fixed elevation:

- Select *Elevation* by clicking the selection button.
- Enter the elevation value in the editable field. To set the elevation of a desired point as a fixed elevation, click the list selection or map selection button and select the point from the list or the map.
- Enter the GPS antenna height in the *Ant Ht* editable field (in the GPS+ case) / the reflector (target) height in the *HR*field (in Optical mode)
- Click *Stake* to start stakeout.
- If required, check the *Create Surface* box. A *Cut/Fill* or *Elevation* model can be generated from 3 or more staked points. The Surface created from Create Surface is saved as TIN file. Click Next to specify the name for a new file. By default it will be *newTIN.TN3*. Click on the Create TIN dialog to start the stakeout.
- If required, check the *Use Alignment* box. The Use Alignment option enables you to use an existing alignment to report stations and offsets. Click Next to select the name of the alignment. Click on the Roads dialog to start the stakeout.
- If required, check the *Define Boundary* box. The Define Boundary option enables you to specify a boundary (polygon) from known points. Click Next to do this in the Surface Boundary dialog.

Surface Boundary

To define a surface boundary:

1. In the **Points/Point List/Linework/Area** field, select the current method of selection of data required to create the boundary.

- For **Points,** in the **Select Pts** drop-down list, choose an option to select multiple points in the job for creating the boundary:

- *All* - all the job points.
- *By Range* - points from a range. More...
- *By Code* - points of a selected code. More...
- *By Code String* - points of a selected code string. More...
- *By Radius* - points around a selected point at a certain distance. More...
- By *Layer* - points on a selected layer. More...
- *From Map* - points selected graphically from the map More...
- *From List* - points from a list of points. More...

- For **Point List**, press the List selection button and select an existing point list in the job. More... The list name will appear in the edit field. The name can also be manually entered into the edit field and added to the point selection by pressing the ok button.
- For **Linework** and **Area**, press either the Map selection button and select an existing linework/area from the map (more...) or press the List selection button and select an existing linework/area from the list (more...). The name will appear in the edit field. The name can also be manually entered into the edit field and added to the point selection by pressing the ok button.

2. The list will display the points currently selected, and the plot will display the created surface boundary.

Icons you can use

MAGNET Field Site general buttons and icons are described here...

Stake Surface Road

To stake a surface of an existing Road:

- Select *Road* by clicking the selection button.
- Click the list selection button and select the name of the road from the list.
- Enter the GPS antenna height in the *Ant Ht* editable field (in the GPS+ case) / the reflector (target) height in the *HR*field (in Optical mode).
- Click *Stake* to start the stakeout.
- If required, check the *Create Surface* box. A *Cut/Fill* or *Elevation* model can be generated from 3 or more staked points. The Surface created from Create Surface is saved as a TIN file. Click Next to specify the name for a new file. By default it will be *newTIN.TN3*. Click the button on the Create TIN dialog to start the stakeout.

Stake Surface

To stake a surface of an existing Surface:

- Select *Surface* by clicking the selection button.
- Click the list selection button and select the file. More...
- Enter the GPS antenna height in the *Ant Ht* editable field (in the GPS+ case) / the reflector (target) height in the *HR*field (in Optical mode).
- Click *Stake* to start stakeout.
- If required, check the *Create Surface* box. A *Cut/Fill* or *Elevation* model can be generated from 3 or more staked points. The Surface created from Create Surface is saved as a TIN file. Click Next to specify the name for a new file. By default it will be *newTIN.TN3*. Click on the Create TIN dialog to start the stakeout.
- If required, check the *Use Alignment* box. The Use Alignment option enables you to use an existing alignment to report stations and offsets. Click Next to select the name of the alignment. Click on the Roads dialog to start the stakeout.

Stake Road

To stake points along the road:

1. Select the **Road**, only **Horizontal** or **Horizontal and VerticalAlignment** to be staked. The names can be entered manually or chosen from the list. The dialog will display the plan of the selection.
2. The **Start** station of the stakeout, the distance from the beginning of the road is displayed.
3. Press the **Transition Points** button to bring up a dialog which will allow you to select which transition points you wish to include in the stakeout. More...
4. For GPS, the **Antenna Height** shows the default height of the antenna reference point (ARP) above the mark. You can edit the antenna type, the value of the antenna height, and the type of height in the Antenna Setup dialog. To do this, click the **Ant Ht** button, and select *Edit*.
5. For TS, in **HR**, enter the reflector height.
6. **Stake Report** shows the name of the current stake report if it is set.
7. Click **Next** to proceed on the Stake Alignment dialog.

For the icons available on the dialog see Icons Description.

Transition Points

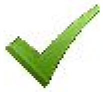
You can select the following **Transition Point Types** to stake:

- **Horz end point:** The node point between horizontal segments. Selected by default.
- **Horz midpoint on curve:** The middle point of a horizontal curve.
- **Vert end point:** The node point between vertical segments.
- **Vert high point:** The highest vertical point.
- **Vert low point:** The lowest vertical point.

To select the point types, use one of the following ways:

1. Click the check boxes near the required types to place the check marks.
2. Use the context menu that depends on the place where it pops up:

- **Select All:** highlights all the types in the list.
- **Select All Below:** highlights all the types below the highlighted line.
- **Select Several**: highlights the required types.
- **Cancel Selection:** removes highlights in the list.
- **Check:** places check marks in the highlighted lines.
- **Uncheck:** clears check marks in the highlighted lines.

Click  to confirm the selection and return to the Stake Road dialog.

Stake Alignment /Slope

This dialog enables you to stake a road without having predefined templates. You can just enter a temporary cross-section.

1. In the **CL Offsets** fields enter the horizontal and vertical offsets to the right (**R**) and to the left (**L**) from the centerline. The vertical offsets can be entered as *Up*, *Down*, or *Grade*.
2. If you select the **Stake Curb/Ditch** check box, you can enter the horizontal and vertical offsets for the curb/ditch. The geometry of the curb/ditch can be *Diagonal*, *Hz/Vert*, or *Vert/Hz*.
3. The plan will show all the entries graphically.
4. Click **Next** to proceed on another Stake Alignment dialog for staking a road and click Stake for staking a slope.

Stake Alignment

The Stake Alignment dialog displays the properties of the cross section on the stakeout station and helps you stakeout all of the desired points.

1. Enter the **Station** where the stakeout is performed. To change the station number by the value of Station Interval, use the arrow buttons or the arrows symbol to switch on/off the keyboard arrow keys.
2. Enter the **Station Interval** of the station increment.
3. Shows the point code of the current segment. The buttons of this field move the current segment point along the cross section. This will reflect on the plan. You can use the arrows symbol to switch on/off the keyboard arrow keys for moving the point.
4. Select between **Right Offset / Left Offset** to enter the horizontal offset from the current segment point as required.
5. Select between **Up Offset / Down Offset** to enter the vertical offset from the current segment point as required.
6. From the drop-down list, select one of the following modes to specify offsets location:
   - *Centerline*: both the horizontal and vertical offset starts at the centerline.
   - *Intersect Left*: the vertical offset starts at the segment point; the horizontal offset starts at the point of intersection of the line parallel to the left segment with the cross-section.
   - *Intersect Right*: the vertical offset starts at the segment point; the horizontal offset starts at the point of intersection of the line parallel to the right segment with the cross-section.

- *Segment*: the horizontal offset starts at the beginning of the segment; the vertical offset starts at the centerline.
- *Surface Left*: the horizontal left offset starts at the beginning of the segment; the vertical offset starts at the point on the surface of the segment that corresponds with the horizontal offset.
- *Surface Right*: the horizontal right offset starts at the beginning of the segment; the vertical offset starts at the point on the surface of the segment that corresponds with the horizontal offset.

7. Click **Stake** to start the stakeout process. More...

Stake Slope

To stake a road's slope:

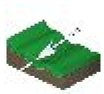
1. Select the **Road**, only **Horizontal** or **Horizontal and VerticalAlignment** to be staked. The names can be entered manually or chosen from the list. The dialog will display the plan of the selection.
2. The **Start** station of the stakeout, the distance from the beginning of the road is displayed.
3. Press the **Transition Points** button to bring up a dialog which will allow you to select which transition points you wish to include in the stakeout. More...
4. For GPS, the **Antenna Height** shows the default height of the antenna reference point (ARP) above the mark. You can edit the antenna type, the value of the antenna height, and the type of height in the Antenna Setup dialog. To do this, click the **Ant Ht** button, and select *Edit*.
5. For TS, in **HR**, enter the reflector height.
6. **Stake Report** shows the name of the current stake report if it is set.
7. Click **Next** to proceed on the Stake Alignment dialog.

For the icons available on the dialog see Icons Description.

Stake Alignment

The Stake Alignment dialog displays the properties of the cross section at the stakeout station and helps you stakeout the catch point (the point where the slope crosses the surface of the terrain) and/or the offset of the catch point.

1. Enter the **Station** where the stakeout is performed. To change the station number by the value of Station Interval, use the arrow buttons or the arrows symbol to switch on/off the keyboard arrow keys.
2. Enter the **Station Interval** of the station increment.
3. Shows the **Hinge Point** code. The hinge point is a point of rotation of the Cut/Fill Slopes. The arrow buttons in this field move the hinge point along the cross section. This will reflect on the plan. You can use the arrows symbol to switch on/off the keyboard arrow keys for moving the point.
4. Select between **Right Offset / Left Offset** to enter the horizontal offset from the current segment point as required.
5. Select between **Up Offset / Down Offset** to enter the vertical offset from the current segment point as required.

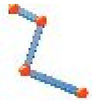
   The icons  /  are used to toggle between the vertical offset and offset perpendicular to the current segment when *Intersect Left* or *Intersect Right* mode is selected.

6. From the drop-down list, select the mode of template offsets:
    - *Auto*: automatically set the last template point (without offsets).
    - *Intersect Left*: the vertical offset starts at the segment point; the horizontal offset starts at the point of intersection of the line parallel to the left segment with the cross-section.
    - *Intersect Right*: the vertical offset starts at the segment point; the horizontal offset starts at the point of intersection of the line parallel to the right segment with the cross-section.
    - *Segment*: the horizontal offset starts at the beginning of the segment; the vertical offset starts at the centerline.
    - *Segment Slope*: it is very similar to the normal *Segment*, but it uses the next point in the direction of the slope to calculate the slope ratio. This mode also skips the Stk Slope dialog, since it has no effect. Click Stake to start the stakeout process.
    - *Surface Left*: the horizontal left offset starts at the beginning of the segment; the vertical offset starts at the point on the surface of the segment that corresponds with the horizontal offset.
    - *Surface Right*: the horizontal right offset starts at the beginning of the segment; the vertical offset starts at the point on the surface of the segment that corresponds with the horizontal offset.
7. Click **Next** to proceed on the Stk Slope dialog.

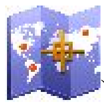
Stk Slope

This dialog shows cut/fill for template slopes, and enables you to edit the cut/fill slope values.

1. Select either the **Template Slopes** radio button to apply the template cut/fill slope values or the **Enter Slopes** radio button and enter the cut/fill slope values as required.
2. Click **Stake** to start the stakeout process. More...

 Stake Linework

To stake a linework:

1. Select **Linework / Code** to toggle through two linework selections:
   - *Code*: allows linework selection by CodeString. Select the code from the drop-down list and a string from the Strings list. The plot of the linework will be displayed.
   - *Linework*: allows linework selection from the job lineworks (listed in the dialog) or a polyline selected from the Map (click ). If the line is a background line, copy the line to the job, update the list of lineworks and select it in the list.
2. The **Start** point of the stakeout, the distance from the beginning of the road is displayed.
3. For GPS, the **Antenna Height** shows the default height of the antenna reference point (ARP) above the mark. You can edit the antenna type, the value of the antenna height, and the type of height in the Antenna Setup dialog. To do this, click the **Ant Ht** button, and select *Edit*.
4. For TS, in **HR**, enter the reflector height.
5. Select the **Include Transition Point** checkbox if the transition point should be included.
6. **Stake Report** shows the name of the current stake report if it is set.
7. Click **Next** to proceed on the Station & Offsets dialog.

Station & Offsets

The Station & Offset dialog allows you to define parameters for staking offset locations.

1. The **Station**/**Real-Time** button allows you to toggle between staking known and unknown staking locations. For Station:
   - Enter the starting station location. The two arrows allow you to decrease or increase the station by the specified interval.
   - Enter the **Interval** for stations.
   - The **Num Segments** field shows the number of segments of the line determined with the given interval. For instance, a value of 3 indicates the line is divided into three equal segments to stake four points.
2. As required, enter the **Right Offset**/**Left Offset** of the stakeout point with respect to the line at the station shown on the Station field.
3. As required, enter the **Up**/**Down** offset with respect to the height of the line at the station.
4. The **Angle** option enables you to stake one of the three possible offset locations when they stakeout to an

angle point:

- *Offset Bk*: the stake location will be on the (back) segment before the angle point.
- *Offset Ahd*: the stake location will be on the (ahead) segment after the angle point.
- *Bisector*: when bisector is selected, the proposed stake location will be on the bisectrix of the angle at the computed offset from the angle point.

5. The **Curve** option enables you to stake one of the four possible locations when they stakeout to a curve point:

   - *Offset*: the stake location will be on the curve at the offset distance from the curve point.
   - *RP*: the radius point.
   - *PI*: the point of intersection of tangents to the curve drawn at the start and end curve points.
   - *MOC*: the point in the middle of the curve.
6. Click **Back** to return to the previous initial dialog if required.
7. Click **Stake** to perform the stakeout. More...

Store Point

Displays information regarding the Stake Point about to be stored and allows you to add or edit information.

Offset from CP

Offset from CP displays information about the offset from the catch point. This dialog appears after the point has been stored.

- **HD Offset from CP to OP**: Horizontal Distance from the Catch Point to the Offset Point.
- **Cut/Fill at OP wrt CP:** Cut/Fill between the Catch Point and the Offset Point.
- **Cut/Fill at CP wrt HP:** Cut/Fill between the Hinge Point and the Catch Point.
- **Cut/Fill Slope (1:n):** Design Slope between the Hinge Point and the Catch Point.
- **Elevation:** Elevation of the Offset Point.
- **Slope Dist from HP to IP**: Slope Distance from the Hinge Point to the Intersection Point.
- **VD Offset from HP to IP**: Vertical Distance from the Hinge Point to the Intersection Point.
- **Intersection Point**: The Intersection Point (IP) is the point between the vertical line at the Hinge point Cut/Fill slope at the Offset Point.

Stk Info

Displays the Stakeout Information for the point being saved.

Select Value

The select value dialog allows you to select which values are to be displayed in the stake dialog. In the dialog, only select the value you want displayed and press the green check mark to exit the dialog. You will be returned to the stakeout dialog, and the value you selected will now be displayed in the desired label.

Apps folder

Click an icon to use an application built in your device:

Take Photo
Starts the Camera application to take pictures.

Web Browser
Starts Internet Explorer to visit a web address you want.

Photo Viewer
Starts the Pictures & Videos application to view the pictures attached to the current job.

Files
Opens File Explorer to manage the files.

3DMC Inbox
Opens your inbox folder where files sent by other SiteLINK 3D clients are stored.

Scan QR Code
Takes a picture of a required QR code and decodes information from it. This icon is available, when an external or internal camera is connected.

Schedule

When the connection with Enterprise project is established, opens the table with list of tasks for the given project.

Timecard

When the connection with Enterprise project is established, opens the table where you can enter the work hours for any uncompleted task of the project.

Magnet News

To view any information from the Magnet RSS file:

- Check the Internet connection.
- From the bottom drop-down list, select the type of information you are interested in.

- Use the arrow buttons to browse.
- Click the **Link** button to open the Internet Explorer with the information as required.

Schedule

When the connection with Enterprise project is established, the table of the dialog displays the list of tasks for the given project. The **Start Day**, **End Day** and **CMP %** (task completion rate in percentage) fields are available for the each task. If the task is set "**Completed**" the task row in the table is grayed and it is not available to log work for it. To edit the work hours and completion percentage for uncompleted project tasks, you can double click on it to open Timecard dialog.

Using the buttons on the bottom part of the table you can set the desired period and the table will show the current status of the tasks for the selected time interval:

- Click the **By day/week/month** button and select the desired time interval from the list.
- To select the desired day/week/month, use "<<" or ">>" .
- Click the **Current day/week/month** button to set the current day/week/month in the table.

Timecard

When the connection with Enterprise project is established, you can log the time you spend on tasks of the current project in the dialog.

- To select the desired week, use "<<" or ">>" . Click the **Current week** button to set the current week in the table.
- For any task of the project you can enter working hours for the day or days. To save the entered values and send it to the Enterprise service, click the **Submit** button.

  Note 1: If you have entered the works hour in the task only, after clicking the **Submit** button, the Enterprise service calculates the completion percentage (**CMP %**) for the given task.

  Note 2: Once you manually edited the **CMP %** column, after clicking the **Submit** button, the Enterprise service will set manually entered task completion percentage and stop automatic calculation of task completion rate. The entered value will be used for the given task.

  Note 3: The task has a completed status, when completion percentage reaches 100%.

  Note 4: You can submit working hours for completed task also.

Localization in MAGNET Field Site

When performing a task of transformation of GPS points coordinates measured in WGS-84/Datum/Grid coordinate system into a Ground coordinate system, we cannot use the algorithms of transformation which are applied to pre-defined or created by the user grids and datums. In this case MAGNET Field Site does not have predefined relation between two coordinate systems. We use Localization to perform coordinate transformation from WGS-84/Datum/Grid to Ground (and vice versa).

Related topics:

- Basic Concept of Localization
- Localization with Stereographic Projection on WGS84
- Localization with Stereographic Projection on any Datum
- Localization with any Predefined or Created Projection

Basic Concept of Localization

Localization in MAGNET Field Site is a calculation of transformation parameters between WGS84/Datum/Grid and a ground coordinate system. To perform localization, you need to have two independent sets of coordinates for the same point or few points in the MAGNET Field Site. An independent set of coordinates is a set of such point coordinates that do not have a relation between each other. To perform localization in MAGNET Field Site, you need to manually select coordinates in the Ground coordinate system ("Known Point") and in the Grid coordinate system or WGS84/Datum coordinate system ("Measured Point") for the same point in the current job. Such a point is called Localization Point.

In MAGNET Field Site, horizontal localization and vertical localization are performed separately.

- Horizontal localizations use two-dimensional conformal transformations. This kind of transformation is also known as a four-parameter similarity transformation (**Rotation**, **Scale**, and two translation parameters (**DX**, **DY**)). To relate the points' geodesic coordinates (measured with GNSS receivers) to local plane coordinates (obtained with total stations, etc.), a map projection is used as an intermediate step.
- Vertical localizations use a three-parameter transformation (one shift (**HO**) and two slopes (**Hx**, **Hy**) to convert between the points'ellipsoidal or orthometric heights and the elevations in the local height system. These three parameters are necessary in order to specify the plane that would adequately model the difference between the local geoid and the WGS84 ellipsoid in the given local area.

You determine how the localization points will use in localization parameters calculation by selecting the corresponding check boxes:

- **Use Horizontal** - the point will use only in horizontal plane localization.
- **Use Vertical** - the point will use only in vertical al plane localization.

- **Use Horizontal** and **Use Vertical** - the point will use in both localizations.

Also you can select the point which will use only in the determination of rotation between two coordinate systems by selecting the corresponding check box **Rotation Only**.

When you check **Use Horizontal** and / or **Use Vertical** you can use one, two, three and more localization points. For this case MAGNET Field Site will calculate a different set of transformation parameters and residuals. The table displays which parameters are calculated in the process of localization, some parameters can be set to zero depending on the used number of the localization points:

In the horizontal plane:

PARAMETERS	ONE Point	TWO Points	THREE Points	FOUR Points
DX,DY- horizontal offset between two coordinate systems	CALC	CALC	CALC	CALC
Scale - combined Scale factor	CALC	CALC	CALC	CALC
Rotation - rotation between two coordinate systems	0	CALC	CALC	CALC
N Residual, E Residual - residuals on the horizontal plane	0	0	CALC	CALC

In the vertical plane:

PARAMETERS	ONE Point	TWO Points	THREE Points	FOUR Points
H0 - vertical offset between two coordinate systems	CALC	CALC	CALC	CALC
Deflection North, Deflection East- Deflection components	0	0	CALC	CALC
Ht Residual - residuals on the vertical plane	0	CALC	0	CALC

When you check **Rotation Only** for a localization point, you need add to localization one or more localization points with **Use Horizontal** or **Use Horizontal** and **Use Vertical** status. For this case MAGNET Field Site will calculate:

- with using localization point(s) with **Use Horizontal** and **Use Vertical** - DX,DY,H0, Rotation ,Scale, Deflection North, Deflection East, NResiduals, EResidual, Ht Residuals,
- with using localization point(s) with **Rotation Only** - Rotation, NResiduals, EResidual, Ht Residuals,

If a geoid model is set in the job, MAGNET Field Site will use this geoid model for calculating orthometric height (elevation).

The current version of MAGNET Field Site enables you to perform the horizontal localization in three ways:

1. Using default (the stereographic) projection on WGS84 (Find out more...).
2. Using default (the stereographic) projection on any datum (Find out more...).
3. Using any predefined or created projection (Find out more...).

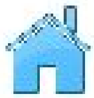
Localization

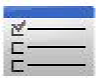
In the dialog you can select the type of localization and see the points are used in the calculation of the transformation parameters.

1. Select the **Type** of Localization. You can see the following types:
   - **WGS-84 ->Local**; For the plane localization will use the stereographic projection on WGS-84. If you perform a task of transformation of WGS-84 coordinates into Ground coordinates for a network where the maximum distance between the local points is less than 5 kilometers, you can use this localization type. Before localization you need to set in the Coordinate System dialog: **Projection** <none>; **Datum** WGS-84.
   - **Datum -> Local**; For the plane localization will use the stereographic projection on selected datum. If you know which datum is used for the local coordinate system, you can use this datum in localization. Before localization you need to set in the Coordinate System dialog: **Projection** <none>; **Datum** ANY_Datum .
   - **Grid->Local**; If you know which projection is used for the local coordinate system, you can use this projection in localization. In this case this projection and corresponding datum will be used in the process of calculation of localization parameters. Such approach to the calculation of the localization parameters between two coordinate systems is more rigorous method, than using the stereographic projection for a unknown local projection. This way allows increasing the distance between localization points (up to some hundred of kilometers depending on the type of the projection) without loss in transformation precision. Before localization you need to set in the Coordinate System dialog: **Projection** ANY_Projection; **Datum** Datum_for_Projection .

2. Click **Add** to open the Add Localization Point dialog for adding control point(s). The quantity of points required for localization depends on how these points are used in localization parameters calculation.

   Note: The localization is recomputed every time a new point is added to the list of localization points. The new coordinate system will be saved under the name "Localization" and is automatically selected in the Coordinate System dialog.

3. Select the **Keep scale 1.000000000** check box to preserve the localization from a scale transformation as required. If this parameter is not selected (default settings), the scale factor will be calculated according to the number of localization points.
4. If needed, click **Edit** to open the **Edit H/V Controls** dialog to change how these points are used in localization parameters calculation.
5. Click **Remove** to remove the highlighted points pair(s).
6. Click **Details** to view the results of the localization.
7. Click  to close the Localization window.

Click  to change Settings for the survey if necessary.

Related task:

- Basic Concept of Localization

The icon opens a pop-up menu which you can select additional features of localization More...

Pop-up menu of Localization

The pop-up menu contains a set of the following options:

- Config Radio
- Edit Points
- **Export to File** opens **To File** dialog, where you can select the desired file format to export the transformation parameters. By default, "gs3" file format is set. The Export to File is enable, when the localization type is **WGS->Local**.
- **Localization Origin** is enable, when the localization type is **WGS->Local** and one or more pairs of points are selected for localization. You can select one of the following:
  - **Center of Localization Points** (default setting) - the geometrical center of pairs of points, which is used in localization, becomes the center of the default map projection
  - **First Localization Point** - the first pair of points, which is used in the localization, becomes the center of default map projection.
  - **Legacy Mode** is enable, when one or more pairs of points are selected for localization. Select it to set the Legacy mode for the horizontal localization. We recommend you to select this mode when you import the Topcon 3D localization file (*.gc3), which was created by all versions of Topcon Tools, TopSURV, Pocket 3D, 3D-Office and versions 1.* of MAGNET Tools and MAGNET Field. If this

mode is not selected (by default), the default mode is applied. We recommend you to select default mode when you import the Topcon 3D localization file (*.gc3), which was created by MAGNET Tools and MAGNET Field from version 2.0. When you import Localization file into the opened job and you click the Use Legacy Localization check box, the Legacy mode is automatically set.

Add Localization Point

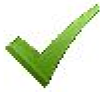
In the dialog you can add point(s) to localization.

1. Select where the localization point will be use:
   - **Rotation Only** - the localization point is used for the determination of rotation between two coordinate systems only. Using one point is not enough to perform localization. Need to add one or more point with "Use Horizontal", or "Use Horizontal"/ "Use Vertical " status. When performing localization the software calculates:
     - the rotation between two coordinate systems using point(s) with "Rotation Only" and point(s) with "Use Horizontal", or "Use Horizontal"/ "Use Vertical "status.

       and

     - the offset and scale between two coordinate systems using point(s) with "Use Horizontal", or "Use Horizontal"/ "Use Vertical "status.
   - **Use Horizontal** - the point is used in the horizontal localization. You can select one, two, three and more localization points in this plane. For each case the application will calculate a different set of transformation parameters and residuals.
   - **Use Vertical** - the point is used in the vertical localization. You can select one, two, three and more localization points in the plane.
2. In the **Known Point** field, enter the *Point* in the local (ground) coordinate system. You can enter the point manually or select it from the map or from the list of the job points.
3. In the **Measured Point** field, enter the *Point* in the coordinate system that is set for the current job. You can enter the point manually, select it from the map or from the list of the job points. For the GPS device type you can measure the coordinates point by clicking . The # field shows the number of the accepted epochs. The parameters of the logging are set through the Settings button. If a point with the same name already exists, the application will open the **Point Check** notification dialog. You can overwrite,

rename, or store the point as a check point. Select the Code for this point and click  to set the point's attributes . More...

4. Click  to save the point and return to the Localization dialog with a newly added point.

Localization details

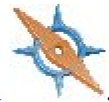
The Localization details dialog displays the input and calculated parameters of the localization:

- *Localize To*: the name of the projection (datum) of the current job to which the localization was applied.
- *Rotation*: the rotation angle between the projection (datum) and local coordinate systems.
- *Scale*: the scale factor between the projection (datum) and local coordinate systems.
- *Offsets*: the offset vector between the origins of coordinate systems.
  North: the horizontal component of the offset vector on the X axis.
  East: the horizontal component of the offset vector on the Y axis.
  HO: the vertical component of the offset vector.
- *Deflections*: the vertical deflection.
  North: the North component.
  East: the East component.
- *Geodetic Origin*: geodetic coordinates of the first localization point from the projection (datum) set in the coordinate system of the current job.
  Lat: the latitude of the first localization point.
  Lon: the longitude of the first localization point.
  Ht: the ellipsoidal height of the first localization point.
- *Local Origin*: local coordinates of the first localization point from the local set in the local coordinate system.
  North: the northing coordinate of the first localization point.
  East: the easting coordinate of the first localization point
  Ht: the orthometric height of the first localization point.

Grid to Ground Transformation

MAGNET Field Site supports two methods for setting the relation between Grid and Ground coordinate systems. One method performs scaling and rotation relative to some point of the job. The other method performs scaling and rotation relative to the origin of the Grid coordinate system.

To get started with transformation:

1. Click the Configure icon  and then the Coordinate System icon .
2. Select the desired projection through which the link will be found with some Ground coordinate system. Only after selecting the projection the option of Grid to Ground transformation will become available.
3. To activate the Grid to Ground (or vice versa) transformation select the Use Grid/Ground check box.
4. Press  to select a desired transformation method (from "Origin Pt", "Avg Job Ht" and "Scale Factor") and set the corresponding parameters for this method.

Find out more:

Creating Ground Projection Relative to a Point

Creating Ground Projection Relative to the Origin of Grid System

Parameters of transformation from the Grid to Ground coordinates can be used in GPS stake of road points. For example, to stake a point at the distance of 100 meters from some point, you have to create a ground coordinate system, which will work as a base coordinate system in GPS stake calculation. To do this, select the corresponding Grid and enter the average height of a desired portion of the road. The created ground coordinate system enables you to stake points at special distances in the grid system.

Creating Ground Projection Relative to a Point

If you know the coordinates of a point in both coordinate systems (Grid and Ground) and also rotation of these systems, then to find the relation between Grid to Ground coordinate systems, select "Origin Pt" from the Parameters list on the Grid/Ground Parameters screen.

This method calculates an offset vector in the horizontal plane between coordinates of a point (called the origin point) in the grid and ground coordinate systems and computes the ground coordinates from the grid coordinates using this offset. If a rotation angle is present between these coordinate systems, MAGNET Field Site can rotate a grid or ground coordinate system relative to this point. Also MAGNET Field Site takes into account a scale factor between these coordinate systems. See the plot here.

After determining the relation between both coordinate systems, MAGNET Field Site will calculate ground coordinates from the grid coordinate system and vice versa.

See an example of using this method below.

Example of using "Origin Pt"

The current MAGNET Field Site job contains two sets of measured points:

- One network of points measured by a GR-3 receiver from a reference station in Grid coordinate system, for example, SPC83 -Ohio(North).

- Another network of points measured by a Topcon Total Station (GTS-220) in the Ground coordinate system with the arbitrary choice of zero BS azimuth.

These networks have:

- A common point - the point CP2 of the GPS network and point CP2_TS of the TS network. This point is the origin point for our transformation.
- A common line - the line between CP2 and CP6 points of the GPS network, and the line between CP2_TS and CP6_TS of the TS network. The azimuths of this line in different networks will be used for calculation of the rotation angle between both coordinate systems.

See Map View for the different networks here.

To perform Grid to Ground transformation for these networks:

1. Click , select the desired Grid system (SPC83 -Ohio(North)) in the Projection field of the Coordinate System screen.
2. Check the Use Grid / Ground box to use this transformation, and press to open the Grid/Ground Parameters screen.
3. Select "Origin Pt" in the Parameters field to activate this method.
4. Select the origin point in the Grid coordinate system (CP2 point) from the list.
5. Remove the coordinates of the selected point in the fields Northing and Easting, which MAGNET Field Site automatically writes in the Grid coordinate system.
6. Type in the values of the Ground coordinates of the origin point (see the Grid/Ground Parameters screen here).
7. Clicking in the Grid/Ground Parameters screen:
   - Calculates a combined scale factor for this point.
   - Combines the GPS point with the TS point. After that, in the Ground coordinate system, the GPS point will have horizontal coordinates of the corresponding TS point, and, in the Grid coordinate system, the TS point will have horizontal coordinates of the corresponding GPS point.
   - Performs grid to ground transformation (and vice versa) taking into account the scale factor.

The Map displays the networks either in the Ground or Grid coordinate system.

To take into account the rotation between these two networks in this transformation:

1. As we see from Map View for the different networks, the grid azimuth is set by direction of the line CP2 - CP6, and the ground azimuth is set by direction of the same line in ground coordinate system (the line CP2_TS - CP6_TS). MAGNET Field Site calculates the corresponding azimuth when you select the start and end

points of the line.

2. Click ... in the Azimuth Rotation field. The Compute Rotation screen allows calculating Grid and Ground azimuth to obtain rotation angle between two coordinate systems (see Grid to Ground Transformation Without Rotation).
3. Click the Compute button in the Ground line. In the Compute Azimuth screen, select the corresponding points (that define the common line in the Ground coordinate system) from the list.
4. Click the Compute button in the Grid line. In the Compute Azimuth screen, select the corresponding points (that define the common line in the Grid coordinate system) from the list.
5. The final rotation angle between two coordinate systems for our example is displayed in the Rotation field. Click ✓ to save this calculation.
6. The Grid/Ground Parameters screen contains all needed values to calculate the relation between the Grid and Ground coordinate systems (see here). Click ✓ in this screen.
7. The Grid to Ground transformation is successfully performed. Find a plot here that displays the error of transformation taking into account the scale factor and rotation for the given example.

Creating Ground Projection Relative to the Origin of Grid System

If you know the value of the scale factor between grid and ground coordinate systems or the average height of the network, then, in the Grid/Ground Parameters screen, you can select "Scale Factor" or "Avg Job Ht".

This method calculates plane ground coordinates by scaling, offsetting, and rotating grid coordinates. Find graphic explaination of transformation only by scaling here, only by ratating here, and only by shifting here.

To find a relation between Grid and Ground coordinate systems:

1. If the scale factor is known, select "Scale factor" in the Grid/Ground Parameters screen, type in a desired value, and select the correct direction (Grid to Ground or Ground to Grid) for this value.
2. If the average height of the network is known, select "Avg.Job Height" in the Grid/Ground Parameters screen and type in the desired value of the height in the Avg.Job Height field.

   The Scale Factor will be automatically calculated from: *Scale_Factor = (1+ Avg.Job Height/Mean_Earth_Radius)*, where *Mean_Earth_Radius* = 6371000.0 m

3. In both cases you can enter the rotation and offsets:
   - If the angle of rotation is known, type in this value into the Azimuth Rotation field. MAGNET Field Site performs rotation of the Ground relative to the origin of the Grid coordinate system.
   - To shift the Ground system from the selected Grid, type in the desired plane offsets on the Northing Offset and Easting Offset fields.

The final coordinates in the Ground will be calculated from: *Nground=Ngrid+Northing_Offset* and *Eground=Egrid+Easting_Offset*