MAGNET Field Layout Help

Version: 3.0

Part Number: 1002704-01

Revision C

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 Layout is a cloud enabled data collection software for quick and accurate construction layout and includes data collection, stakeout functionality and computations.

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 Layout, the application will need to be activated again.
- If the activation is cancelled, MAGNET Field Layout 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 Field Layout 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 Layout 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 Layout 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 Layout startup, the **Connections** dialog runs with the current job selections.

Home screen

See the **Introduction** for general information on MAGNET Field Layout.

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

Job

Manages jobs.

Points

Opens dialog contains the list of the points stored in the application database

Points List

Opens dialog contains a list of existing Point Lists, and the two windows, which display the general view of the selected list in the horizontal and vertical planes

Reports

Edits field and deviation reports.

Enter Plan

This function allows you to draw a plan consists of the elements which are defined by points, segments and arcs.

Calculate

Calculates various coordinate geometry tasks.

Plan

Shows the current job data on the plan.

Connect

Switches between GPS and optical instruments, and sets connections with the instrument, a network, and with MAGNET Enterprise. Not available for onboard.

Setup

Sets up surveying: GPS+ survey or Optical (Total Station) survey. Unavailable for Level configuration style.

As-built

Opens the general dialog for topo survey for GPS+ survey or Optical (Total Station) survey.

Layout
Stakes different objects.

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

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. Not available for onboard.

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

Indicates the connection status with the device. Not available for onboard. You can connect/reconnect with the device in the **Connections** dialog. If the device is disconnected, the indicator is gray. When connecting, it turns yellow, then green, and finally turns into the image of the connected device:

-  - GPS type instrument is activated,

-  - GPS type instrument which obtains correction information.
-  - Optical type instrument is activated.
-  - Hybrid Positioning is activated.

GPS type instrument is the current device type when **Hybrid Positioning mode** is activated. To select the Optical type, click the button.

Optical type instrument is the current device type when **Hybrid Positioning mode** is activated. To select the GPS type, click the button.

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 Layout 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** and **Schedule** dialogs.
- Click *Configure Menus* to show / hide functions in the menus. Find out **more...**
- Click *Activate Modules* to perform a licence 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 Layout. Find

out **more..**.

- Click *About* to view basic information about MAGNET Field Layout 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 Layout 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 Layout. 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 Layout.

About MAGNET Field Layout

This lets you:

- View information about the current version of MAGNET Field Layout.
- 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.

Job Folder

Jobs include all data collected in surveys of different configurations. Upon installation, MAGNET Field Layout 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

Opens the Configure menu to select desired configuration of the current job.

Exchange

Opens the exchange menu where you can exchanges data between an active job and other jobs, files and enterprise projects.

Layers

Opens the dialog where you can edit the existing layer or create a new layer with custom style.

Polylines

Opens dialog with the existing polylines. You can edit any polyline from the list or create a new polyline.

Areas

Opens dialog with the existing areas. You can edit any area from the list or create a new area.

Codes

Opens 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.

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 Layout always opens the last used job after the product activation and **Connections** are performed.

All existing jobs that are created/opened with MAGNET Field Layout 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 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

Current Job: shows the name of the current 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 Layout 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 Layout 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

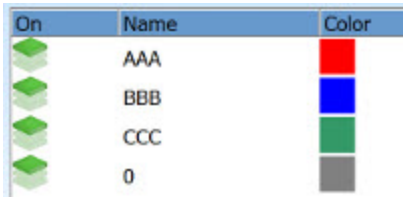
2. Select the **Type** of files from the drop-down list to display the files in the list for the current folder.
3. Enter the **Name** of the file to select or select the desired file from the list.

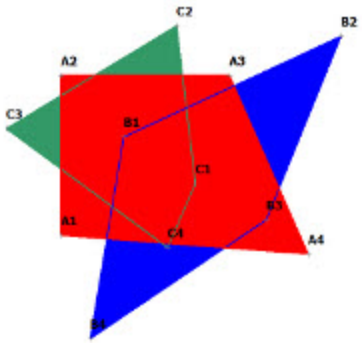
 Layers

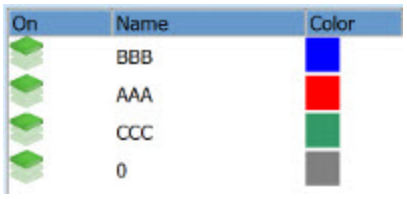
The *Layers* dialog contains a list of layers in the job, and a set of tools for editing. The list displays the name, color and status of each layer.

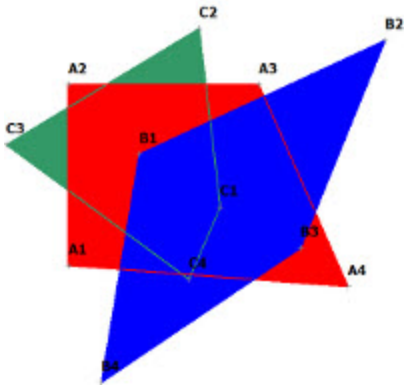
To edit a layer, highlight it and use necessary tools:

- Use either  or button to make the selected layer visible/invisible.
- To change the color for the objects in this layer, click a square in the *Color* column and select a desired color in the *Select Color* dialog.
- To sort the layer by *Name*, or *Color*, or *Status*, click the corresponding column's title. The existing layers will be arranged by this parameter in the dialog. During layers sorting the  and  will be disabled, but the **View** button will be enabled.
- To sort the layer in order to display on the *Map*, click the **View** button and use either  or  button to move the selected layer up or down in the ordering. This option is useful when the job contains the overlapped objects:









- To change the color for the objects in this layer, click a square in the *Color* column and select a desired color in the *Select Color* dialog.
- Click **Add** to create a new layer. After clicking the button, the **Add Layer** dialog appears. The dialog contains three tabs, where you can enter all needed features for the new layer:
  - **Layer tab**
  - **Style tab**

    - **Avoidance tab**
- Click **Edit** to change the properties of the highlighted layer. After clicking the button, the **Edit Layer** dialog appears. The dialog contains four tabs, where you can edit all needed features for the existing layer:
    - **Layer tab**
    - **Style tab**
    - **Objects tab**
    - **Avoidance tab**
- Click **Delete** to delete the highlighted layer from the list. The default layer cannot be deleted. You can delete a layer which does not have any objects.
- Right clicking on a layer opens the context menu:
    - **Check**: places the check mark in the **Visible** check box to make this layer visible. In this case, the icon  stands in the **On** column.
    - **Uncheck**: clears check mark in the **Visible** check box to make this layer invisible. In this case, the icon stands in the **On** column.
    - **Select All**: highlights all the layers in the list.
    - **Select All Below**: highlights all the layers below the highlighted layer.
    - **Select Multiple**: highlights any selected layer.
    - **Cancel Selection**: removes highlights in the list.

Note: If highlighting multiple layers, you can check / uncheck or delete all these layers.

Layer tab

The *Layer* tab allows you to edit the selected layer's information or create a new layer:

1. In **Layer Name**, enter a desired name of the layer.
2. Select the **Visible** check box to make this layer visible/invisible. The icon  stands to mark the visible layer.
3. Enter a **Note** for the layer.

Style tab

The *Style* tab allows you to edit the style for drawing points, lines and areas to be displayed in this layer:

1. In **Point Type** panel, select an icon for the points from the drop-down list and check its display.
2. In **Line Style**, select a style for the lines and the *Width* of lines in points.
3. In **Area Style**, select a fill style and **Transparency** for the area.
4. Click **Color** to select the color for the objects in this layer.

Objects tab

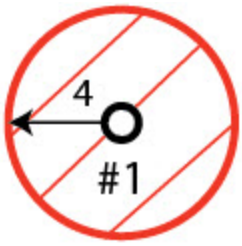
This tab will display if you open the existing layer for editing. Expand the tree nodes to see the names of the objects (Points, Lines, Areas and Roads) belonging to the selected layer.

Avoidance tab

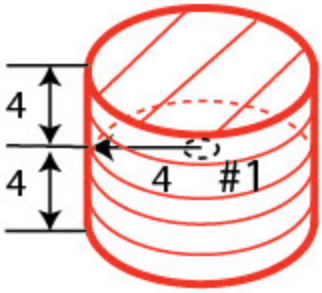
The *Avoidance* tab allows you to create a layer for an avoidance zone. The  blinking notification displays in *Topo* / *Autotopo* / *Stake* dialog, if a TS or a GPS measurement is performed in this area.

- In the **Avoidance mode** you can select for which planes the option will be used for point, linework and area:
  - select **OFF** to switch the **Avoidance mode** off;
  - select **2D** for working in the horizontal plane only;
  - select **3D** for working in the horizontal and vertical planes.
- In the **Proximity** field, specify the size of a boundary area for point, linework, area or surface. The blinking notification will be displayed when a measurement is performed within the boundary area or avoidance area. This boundary will be defined by the distance from the object:
  - for point:

2D mode



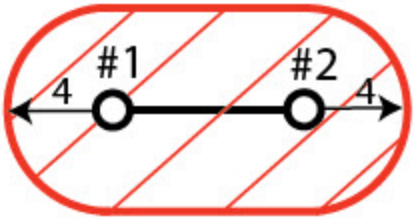
3D mode

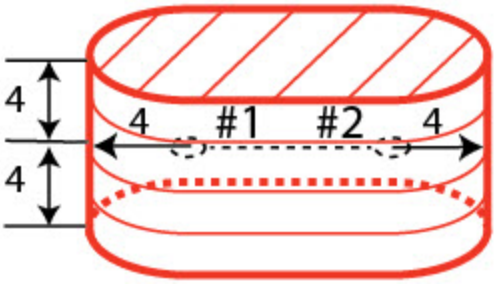


  - for a linework:

2D mode

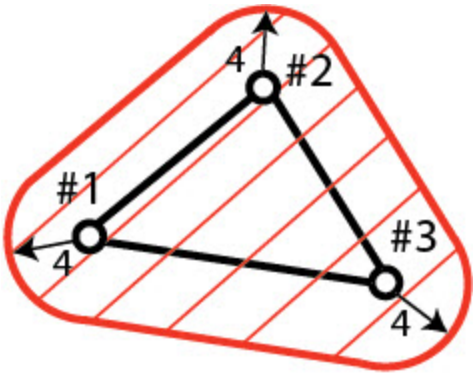
3D mode



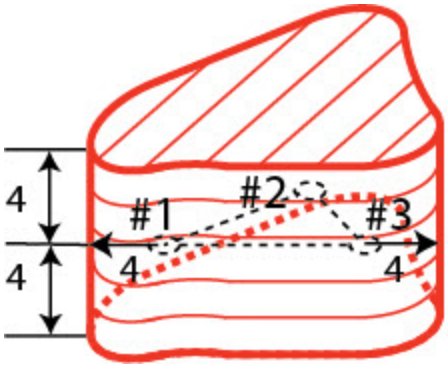


- for area:

2D mode



3D mode



where:

- #1 ; #2 ; #3 - points,
- #1 - #2; #2 - #3 - linework,
- 4 - proximity,
-  - boundary and avoidance areas.

Note: If you do not enter any changes in the tab and close it, the **Avoidance** check box will be automatically unselected in the **Layer tab** .

Edit Polyline

The Polilyne dialog contains a list of existing Lineworks, and the two windows, that present the general view of the selected linework in the horizontal and vertical planes. To view the current selected linework 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 linework:
  Points in Line tab
  Layer/Style tab
- Click **Add** to create a new linework.
- Click **Delete** to delete the selected line from the list.

Points in Line tab

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 Plan* enables you to set such a control code.
5. From the **Select 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 Plan* - 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 tab

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 Plan
- 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...**

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 Area

Contains a list of existing areas and the two windows that represent the view of the selected area in the horizontal and vertical planes. To view the current selected area in a larger map, double-click one of the map plots.

- Click **Edit** to edit the properties of the selected area:
  Points in Area
  Layer/Style

- Click **Add** to create a new area.
- Click **Delete** to remove the selected area from the list.

Points in Area tab

Displays a list of existing points in the selected Area and the general view of the area boundary. Area is bound with a closed line. Line nodes (points) have the same code of area type to form an area boundary. The point highlighted in the list of points will be marked with a yellow circle on the Plan view.

To edit the area:

1. If required, change the **Name** of the area.
2. To delete a point from the area, 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 Plan* enables you to set such a control code.
5. From the **Select 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 Plan* - 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 to arrange an area. **More...**
    - *From List* - points from a list of points. **More...**
    - *From Line* - all points from a linework. **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.

Layer/Style tab

To edit the area properties to display the line and existing points in the selected area 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. In ***Area Style***, select a fill style and transparency for the area.
5. Click **Color** to choose the color for the point, the line and the area.

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.

Configure folder

Click an icon to configure the settings:

Equipment

Creates or edits a survey configuration.

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.

Instrument

Sets the temperature and air pressure for total stations only.

Display

Customizes the interface to display data in the job.

Alarms

Sets alarm parameters.

Codes

Sets code global parameters.

Deviation Reports

Configures reports for staking.

Enterprise

Configures the user account to enter into the MAGNET Enterprise. Not available for onboard.

Select Survey Configuration

Survey Configuration is a set of parameters that control a survey, define communication between devices, measure and store points. MAGNET Field Layout 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.

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 mode** is available for selection when using any GPS+ continuous and only Robotic optical configurations for one job. Hybrid positioning 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 Layout provides the set of independent wizards for creating configurations for the following GPS+ survey types:

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

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.

8. 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.

9. Network RTK: As-built Survey

In the dialog 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 needed, 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 **As-built** dialog in GPS+ mode.

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

10. Network RTK: Layout 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/Direction*- 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.

11. Network RTK: Layout Survey

In the dialog 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 needed, 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 **Layout** dialog in GPS+ mode.

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

12. Network RTK: Layout 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 Layout automatically generates one note for each stakeout point: Sta5+5.5R5.0, Cha505.5R5.0, or No.5+5.5R5.0 respectively.

13. Network RTK: Tracking

Select one of the following check boxes 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.

13. 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.

14. 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.

4. 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.

5. 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.

6. 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.

7. 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.

8. RTK: As-built Survey

In the dialog 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 needed, 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 **As-built** dialog in GPS+ mode.

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

9. RTK: Layout 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 **Normal View 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/Direction*- 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.

10. RTK: Layout Survey

In the dialog 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 needed, 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 **Layout** dialog in GPS+ mode.

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

11. RTK: Layout 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.

12. 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.

13. 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.

14. 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.

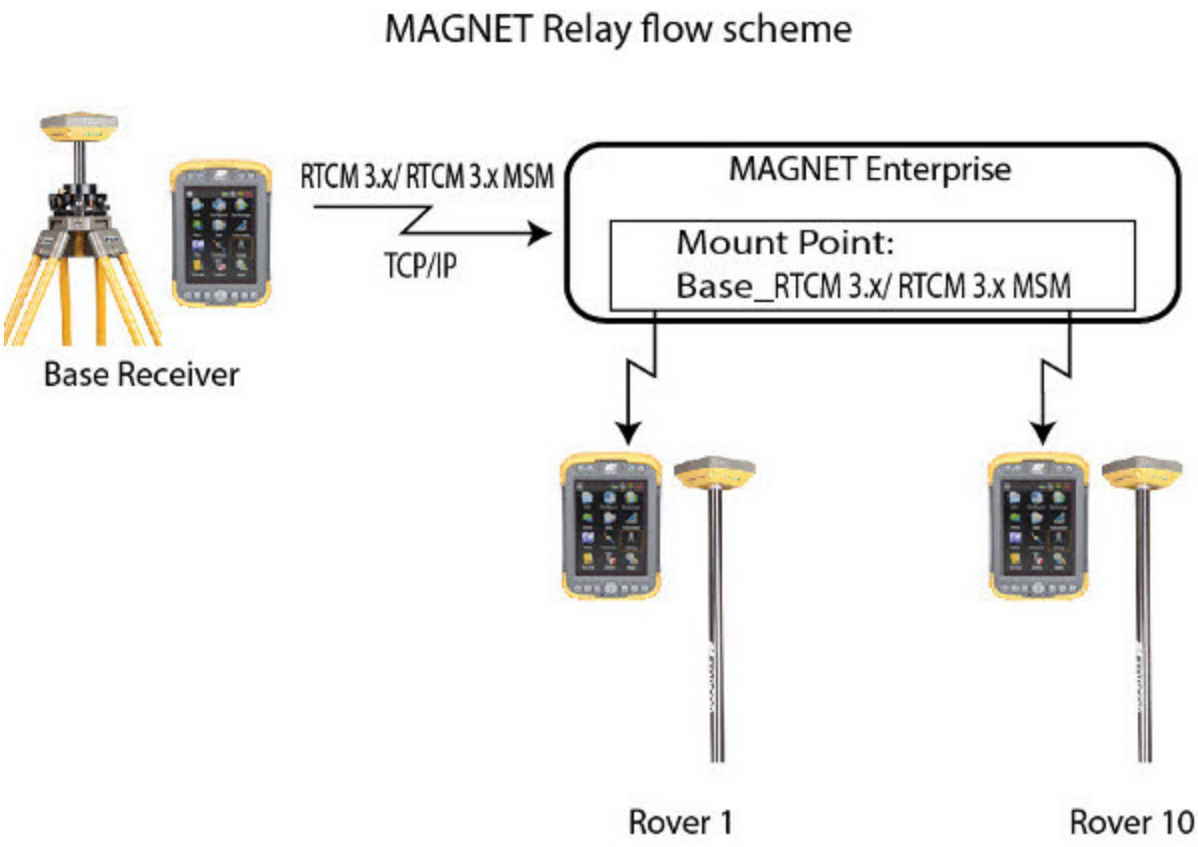
MAGNET Relay

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 Layout.

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 Layout 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.

5. 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.

6. 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.

7. 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.

8. 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.

9. 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.

.

10. MAGNET Relay: As-built Survey

In the dialog 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 needed, 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 **As-built** dialog in GPS+ mode.

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

11. MAGNET Relay: Layout 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/Direction*- 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.

12. MAGNET Relay: Layout Survey

In the dialog 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 needed, 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 **Layout** dialog in GPS+ mode.

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

13. MAGNET Relay: Layout 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 Layout automatically generates one note for each stakeout point: Sta5+5.5R5.0, Cha505.5R5.0, or No.5+5.5R5.0 respectively.

14. MAGNET Relay: Tracking

Select one of the following check boxes 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.

15. 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.

16. 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**.

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.

4. 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.

5. 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.

6. 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.

7. 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.

8. Real Time DGPS/NMEA: As-built Survey

In the dialog you can set the following parameters:

1. **Solution**: for this 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 needed, 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 **As-built** dialog in GPS+ mode.

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

9. Real Time DGPS/NMEA: Layout 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/Direction*- 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.

10. Real Time DGPS/NMEA: Lauout Survey

In the dialog you can set the following parameters:

1. **Solution**: for this 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 needed, 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 **Layout** dialog in GPS+ mode.

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

11. 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 Layout automatically generates one note for each stakeout point: Sta5+5.5R5.0, Cha505.5R5.0, or No.5+5.5R5.0 respectively.

12. 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.

13. 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.

14. 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.

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. Optionally, you can:
   - to use relative antenna calibration for the **base**(the default is the use of absolute calibrations). To do this, select the **Use relative calibration** option from the pop-up menu that displays by clicking  in the top left corner. This selection is used when the base transmits data correction to a rover with the relative calibration.
   - to correct calculate the height of the ground point, where the **rover** is installed. To do this, select the **Relative calibrations on Base** option from the pop-up menu that displays by clicking  in the top left corner. This selection is only used when the relative calibration is selected for the corresponding base receiver.

     Note: This option is available for *Network RTK* and *RTK* types.
5. Click  to save the settings.

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:

- **RTK**
- **Network 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**

Internet Address

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

- **Network RTK**
- **RTK: for Base receiver**

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.

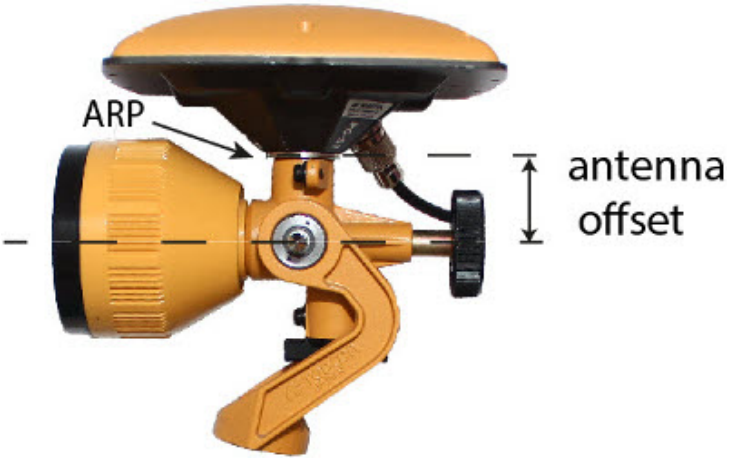
When you activate the Hybrid Positioning, you can:

1. Perform reference points setup 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 **Reference Points** 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 Plan Properties dialog. Select the check box and click  to load Bing Maps to the job.

Reference Points Setup in Hybrid Positioning

Before performing reference points setup you need activate **Hybrid Positioning**, select desired prism and select **Hybrid positioning antenna offset** check box and open **Reference Points** dialog.

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...**

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.
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.

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 Layout 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.

Units

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

- **Distance tab** (including area and volume)
- **Angle tab**
- **Coordinate tab**
- **Other tab**

Distance tab

On the 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

Angle tab

On the 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...**

Coordinate tab

On the 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...**

Other tab

On the 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.

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.

Display

You can configure the display of the following settings:

- **Coordinate Type** to view coordinates for the coordinate system type selected
- **Coordinate Order** to display: Northing/Easting/Height or Easting/Northing/Height
- **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** select a desired format to display stationing the centerline:
  1234.000 - as Chainage, the distance to the station along the centerline

12+34.000 - as US Station (American format), when using 100 units as a full station plus a remainder
1+234.000 - as EU Station (Euro format), when using 1000 units as a full station plus a remainder
1/234.000 - as SWE Station (Swedish format), when using 1000 units as a full station and a slash before a remainder

Alarms

In the dialog you can configure alert conditions. The dialog contains four tabs:

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

1. On the **Main** tab you can select:
   - **Audible Alarm** to enable audible alarms. The alert will sound automatically when an alert situation occurs.
   - **Enterprise Alarm** to enable a sound when new chat comes.
   - **RTCM.3.x Coordinate Systems** to enable a warning that the coordinate system is set by reading the RTCM messages. This option will work when a GPS device is selected.
   - **Avoidance Alarm** to enable the  blinking notification in *Topo* / *Autotopo* / *Stake* dialog, if a TS or a GPS measurement is performed in this area.
2. On the **Controller** tab you can select:
   - **Power Alarm** to enable a sound when conditions of low power of a controller are appeared.
   - **Memory Alarm** to enable a sound when conditions of low memory of a controller are appeared.
3. On the **GPS+** tab you can select:
   - **Power Alarm** to enable a sound when conditions of low power of a receiver are appeared.
   - **Memory Alarm** to enable a sound when conditions of low memory of a receiver are appeared.
   - **Radio Link** to enable a sound when poor radio link is detected.
   - **Fix-Float** to enable a sound when Float solution is set after Fixed solution.
   - **Base Changed** to enable a sound when the rover began to receive a correction data from other real or virtual base station.
4. On the **Optical** tab you can select:
   - **Power Alarm** to enable a sound when conditions of low power of an optical device are appeared.
   - **Track** to enable a sound when conditions of loss of the target in automatic tracking mode for optical instruments are appeared.
   - **Grid/Ground Warning** to enable a warning when a grid coordinate system is set.
5. Click  to observe information about the controller and the connected device status.

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 Plan 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 Layout 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 **Allow 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

Deviation 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.

Enterprise Configure

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 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. Not available for onboard.

Enterprise Download
Downloads data from MAGNET Enterprise. Not available for onboard.

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 grayed 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 grayed 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 Layout 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 Layout 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 Layout 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 Layout 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 Layout 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 Layout 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 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 Layout 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 multiple 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 Layout 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** 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 **Load as background** 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 **Import text** to import text from DXF/DWG files.
4. Select **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.

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.

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,

  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

 backsight

 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 or view or create:

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 linework 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 or Area 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 Layout 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 Layout 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

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 preview and edit the image when you capture it.

- The general editing functions include Zoom In, Zoom Out, Show All.
- To set the image **Size**, select dimension from the drop-down list.
- To geotag the image, click  . **More...**
- Click  to capture the image and show the **Image Preview**.

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.

Add Point

In the dialog you can create a new point. Enter a name, coordinates; select a code, layer and point style; and attach the corresponding image by using the following tabs:

- **Point tab**
- **Layer/Style tab**
- **Image tab**

Point Attributes

The dialog allows you to set attributes for the point being created or edited in the following tabs:

- **Code Attributes tab**
- **Layer tab**

- Photo tab
- Note tab

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** (**Ctrl Code**)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.

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.

Note tab

The tab is used to enter additional information. Type the text in the Note field.

Edit Point Lists

The List of Point Lists dialog contains a list of existing Point Lists, and the two windows, which display the general view of the selected list in the horizontal and vertical planes. To view the currently selected point list in a larger map, double-click one of the map plots.

- Click **Edit** to edit the properties of the selected point list:
  Point List tab
- Click **Add** to create a new point list.
- Click **Delete** to remove the selected point list from the list.
- Click **Copy** to create a copy of the selected point list with a new name.

Point List tab

Displays a list and the map view of existing points in the selected Point List. The point highlighted in the list of points will be marked with a yellow circle on the map view.

To edit the point list:

1. If required, change the **Name** of the point list.
2. To delete a point from the point list, 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. From the **Select Points** drop-down list, choose an option to select a point(s) from the job to add to the end of the point list.
   - *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 Plan* - 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...**
5. 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.

Edit Deviation Reports

The Deviation 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.

Deviation 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 Deviation Report

Displays design references and appropriate information for this report type.

Enter Plan

This function allows you to draw a plan consists of the elements which are defined by points, segments and arcs. All created objects will be displayed in the graphical view in the **Enter Plan** dialog. You may only create these objects, and you cannot edit them.

Note: You may edit objects, created in the **Enter Plan** dialog by using the **Points** and the **Edit Line** dialogs.

Fields for defining new objects parameters are located at the bottom of the dialog. Field type depends on the current mode. To set current mode, click  and select the required mode from the list.

If the ***Draw Line*** is selected, lines or arcs can be created and displayed after configuring the required mode:

- ***Length*** – in this mode you may define the line by its length and direction in the appropriate fields from a start point. You may select the start point in the field () from the list or by clicking on the plan.
- ***Point*** – in this mode you can create a line from one existing point to another existing point. The additional field for end point of the line () is displayed in this mode. You may select the required points from the list or by clicking on the plan.
- ***Arc*** – in this mode you may create an arc with the right or left turning from a start point.
- ***Arc 2 pt*** – in this mode you may create an arc, defined by two existing points.
- ***Arc 3 pt*** – in this mode you may create an arc, defined by three existing points.
- ***Rectangle*** – this mode allows automatic fourth point creation to complete a rectangle.
- ***Close*** – this mode allows automatic closing of a linework. This mode is available only if the linework contains at least two lines segment.
- ***Restart*** - this mode finishes the creating of the linework or point.

If the **Draw Line** is unselected, points can be created and displayed after configuring the required mode:

- ***Length*** – in this mode you create an end point of a line. This line will be defined by its length and direction angle from a start point, which may be specified in the appropriate fields. You may select the start point in the field () from the list or by clicking on the plan.
- ***Arc*** - in this mode you create an end point of an arc.
- ***Restart*** – this mode finishes the creating of the linework or point.

Fields and buttons of the **Enter Plan** dialog:

Defines the start point of a line/arc or the reference point for another point creation.

Defines the end point of the line/arc. This field available in the ***Point*** and ***Arc 3 pt*** mode.

Defines the center or third point of an arc. This field available only in the ***Arc 3 pt*** mode.

Opens the list of the points in the job. You may select any point from this list.

Defines the direction. This field is displayed in all modes, but it is available for the parameters input only in the ***Length*** and ***Arc*** modes.

Defines the line length. This field available only in the ***Length*** mode.

Defines the arc radius. This field is available only in the ***Arc*** and ***Arc 2pt*** modes. This field contains the list of the following parameters, which may define the arc radius:

- Radius (default)
- Chord angle
- Curve angle

Click the button to see the list and select the required value.

Defines the arc delta angle. This field available only in the ***Arc*** mode. This field contains the list of the following parameters, which may define the arc length:

- Delta angle (default)
- Length of the curve
- Length of the chord
- Length of the tangent
- Middle ordinate (the distance from the midpoint of a chord to the midpoint of the corresponding curve)
- External (the distance from the midpoint of the curve to the intersection point of tangents).

Click the button to see the list and select the required value.

Allows quickly add or subtract standard angles to the current direction. Extend function sets the value angle of tangent direction to the previous element in the linework.

 Measures the direction and distance between two points on the plan.

 Clicking on the button opens a list of the modes. You can set the current mode.

 This mode discards the last performed action. It will be unable if the creation of an object is completed.

 Add a new element to the current linework. The end point of the newly created object is automatically set as the new start point for next element.

Also the **drawing** toolbar are enabled in the **Enter Plan** dialog.

Creating points

1. Click  and unselect **Draw Line** mode. In this mode you can create a point as an end point of a line or an end point of an arc.
2. To create an end point of a line, click  and select ***Length***. To create an end point of an arc, click  and select ***Arc*** (Arc Right or Arc Left).
3. Enter the start point name in the field  or select from the list  .
4. For point of a line:
   - enter the desired direction in the field  and length in the field  .

   For point of a curve:
   - enter either radius of arc or one of the two parameters unambiguously defining the radius: chord angle, or curve angle in the field  .
   - enter the delta (the angle between the radii corresponding to the curve), or one of five parameters unambiguously defining the curve length: length of the curve, length of the chord, length of the tangent, middle ordinate (the distance from the midpoint of a chord to the midpoint of the corresponding

curve), external (the distance from the midpoint of the curve to the intersection point of tangents) in the field  .

- enter the azimuth of the tangent to the start point of the arc in the field  .

5. Click the button  to create the point.

6. The **Plan** and **Points** dialogs will show the point(s).

Creating lines by using two points

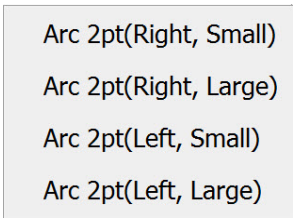
1. Click  and select **Draw Line** mode. In this mode you can create a line and arc.
2. Click  and select ***Point***.
3. Enter the start point name of the line in the field  or select from the list (click ).
4. Enter the end point name of the line in the field  or select from the list (click ).
5. Click  to draw the line and to continue creating of the linework from the end points of the previous line. To finish the creating of line(s) select ***Restart***.
6. The **Plan** and **Points** dialogs will show the point(s).

Creating Arcs

1. Click  and select **Draw Line** mode. In this mode you can create a line and arc.
2. Click  and select ***Arc*** (***Arc Right*** or ***Arc Left***).
3. Enter the start point name in the field  or select from the list (click ).

4. Enter the radius of the arc, or one of the two parameters unambiguously defining the radius: chord angle, or curve angle in the field  .
5. Enter the delta (the angle between the radii corresponding to the curve), or one of five parameters unambiguously defining the curve length: length of the curve, length of the chord, length of the tangent, middle ordinate (the distance from the midpoint of a chord to the midpoint of the corresponding curve), external (the distance from the midpoint of the curve to the intersection point of tangents) in the field  .
6. Enter the azimuth of the tangent to the start point of the arc in the field  .
7. Click  to draw the arc and to continue creating the linework from the end points of the arc. To finish the creating of arc(s) select ***Restart***.
8. The **Plan** and **Points** dialogs will show the point(s).

Creating Arcs by two points

1. Click  and select **Draw Line** mode. In this mode you can create a line and arc.
2. Click  and select ***Arc 2pt***: 

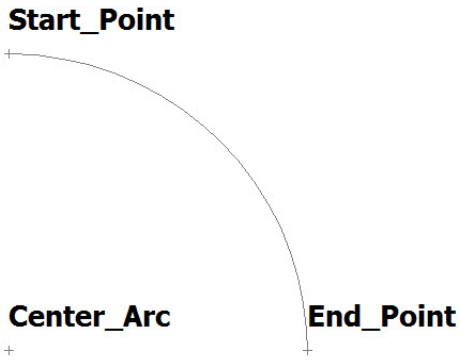
3. Enter the start point name of the arc in the field  or select from the list (click ).
4. Enter the end point name of the arc in the field  or select from the list (click ).
5. Enter the radius of the arc, or one of the two parameters unambiguously defining the radius: chord angle, or curve angle in the field  .
6. Click  to draw the arc and to continue creating the linework from the end points of the arc. To finish

the creating of arc(s) select ***Restart***.

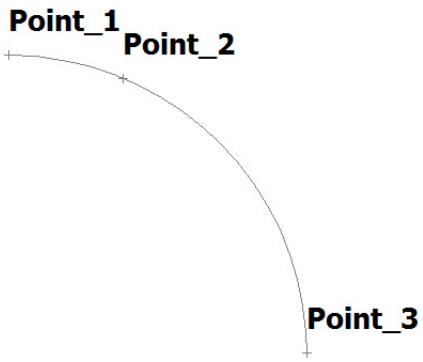
7. The **Plan** and **Points** dialogs will show the point(s).

Creating an Arcs by three points

1. Click  and select **Draw Line** mode. In this mode you can create a line and arc.
2. Click the button and select ***Arc 3pt***:
   - One point as center of the arc and two points as start and end points of the arc (for ***Arc 3pt (RP,Small)*** and ***Arc 3pt (RP,Large)*** modes.) See picture below for details:



   - All three points are located on the curve (***for Arc 3pt(PC)***). See picture below for details:



3. Enter the start point name of the arc in the field  or select from the list (click ).
4. Enter the end point name of the arc in the field  or select from the list (click ).

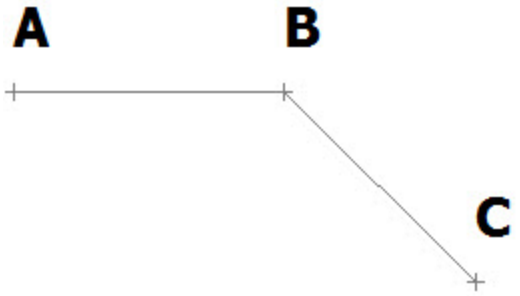
5. Enter the name of the arc center point (for ***Arc 3pt (RP,Small)*** and ***Arc 3pt (RP,Large)*** modes) or third point of the arc (for ***Arc 3pt (RP,Large) mode)*** in the field  or select from the list (click ).
6. Click  to draw the arc and to continue creating the linework from the end points of the arc. To finish the creating of arc(s) select ***Restart***.
7. The **Plan** and **Points** dialogs will show the point(s).

Creating Rectangles

To activate the ***Rectangle*** mode need to create a linework which contains two segments.

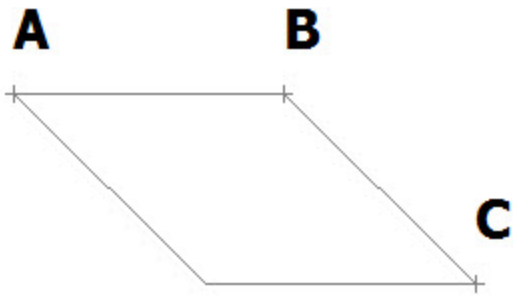
1. Creating a linework with two lines segment:
   - Click  and select **Draw Line** mode. In this mode you can create a line and arc.
   - Click  and select ***Length***.
   - Enter the name of the start point in the field  or select from the list (click ).
   - Enter the desired direction in the field  and length in the field  for creating first segment.
   - Click  to draw the line.
   - Enter the desired direction in the field  and length in the field  for creating second segment.

- Click  to draw the line:



2. Click  and select ***Rectangle*** mode.

3. Click  to create a rectangle:

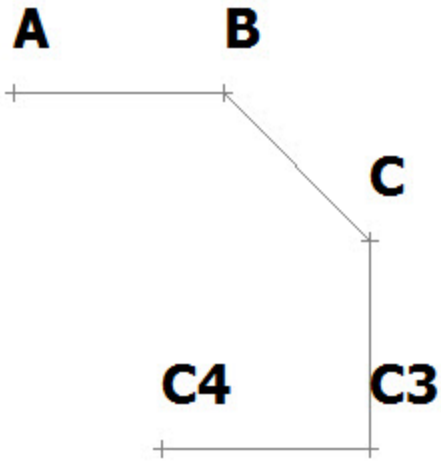


Creating Close figures

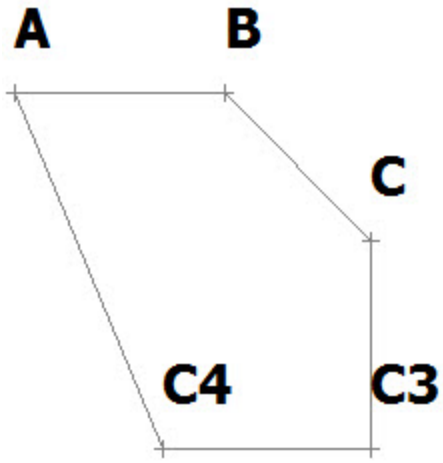
To activate the ***Close*** mode need to create a linework which contains no less than two lines segment.

1. Creating a linework with two lines segment and more :
   - Click  and select **Draw Line** mode. In this mode you can create a line and arc.
   - Click  and select ***Length***.
   - Enter the name of the start point in the field  or select from the list (click ).
   - Enter the desired direction in the field  and length in the field  for creating first segment.

- Click  to draw the line.
- Enter the desired direction in the field  and length in the field  for creating second segment.
- Click  to draw the line.
- You can create more segments:



2. Click  and select ***Close*** mode.
3. Click  to create a close figure:



Distance measurement between two points

To measure distance between two existing points of the job:

1. Click on the field  to activate the button  .
2. Click  .
3. On the plan select first and second points.
4. The field  displays the calculated distance between these points.

Azimuth measurement of direction from point to point

To measure the direction azimuth of for two existing points:

1. Click on the field  to activate the button  .
2. Click  .
3. On the plan select first and second points.
4. The field  displays the calculated azimuth of direction for these points.

Angle measurement for three points

To measure the angle for three existing points:

1. Click  and select **Draw Line** mode.
2. Click  and select ***Arc (Arc Right or Arc Left)***.
3. Click on the field  to activate the button  .

4. Click  .
5. On the plan select first, second and third points.
6. The filed  displays the calculated angle for these points,

Calculate folder

Click an icon to calculate the task:

Length

Computes the inverse (azimuth and distance) between two known points.

Pt to Line

Computes the station of a known point inverse to a known line.

Pt to Arc

Calculates the station of the known point inverse to the known curve.

Pt to Polyline

Computes the station of a known point inverse to a known polyline.

Direction Point

Calculates the coordinates of a point, using a known point, and angle and distance offsets from the known point.

Polyline Report

Computes the inverse for all the vertices in a linework with respect to a known point.

Arcs

Computes a full set of parameters to determine a curve using four various minimum sets of specified parameters.

Corner Angle

Calculates the angle between two lines, a start line and an end line, which have a common mid point.

Offsets

Calculates the coordinates of points along a line, a corners, a arc, a polyline or multiple points

Area

Computes the area of a polygon with known vertices.

Triangle

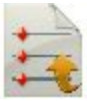
Calculates the area and all parameters of a triangle by three given parameters.

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.

 Length

To calculate the azimuth and the distance (inverse) 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 percentage terms.
   Slope distance: the computed distance between the two points.
4. The **Plan** 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 **Point** 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:

     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. The "-" sign means that the known point is lower than the calculated point.
   - Click  to save the calculated point to the points list.
   - Click  to save the data to a txt file if required.
3. The **Plane** 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** 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 calculated point in the job or/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 Sta: the starting station of the curve.
   Sta 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. The "-" sign means that the known point is lower than the calculated point.

3. The **Plane** tab shows the illustration for the results.

Inverse Point to Linework

To calculate a known point inverse to a known linework:

1. In the **Input** tab:
   - Select the known **Point** name. The point can be entered manually or selected from the map  or from the list  of the job points.
   - Choose either **Linework** or **Code** for selecting the linework.
     - *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's 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 from the list.
   - The **Start** station and the horizontal plan of the selected linework will be shown.
   - The **Pt** field will display the default name of the known point's projection on the linework. 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:
     Projected Point: the name of the known point's projection on the linework.
     North, East, Elev: the coordinates of the projected point.
     Point: the name of the known point.
     Linework: the name of the linework.
     Tangent Azimuth: the azimuth of the linework segment at the projection point.
     Projected Azimuth: the azimuth of the perpendicular from the known point to the linework.
     Start Station: the starting station of the linework.
     Station: indicates the distance between the starting point and the projection of the known point

on the linework.
Offset: the horizontal offset between the known point and the projected point.
dHeight: the vertical offset between the known point and the projected point. The "-" sign means that the known point is lower than the calculated point.

- Click  to save the calculated point to the points list.
- Click  to save the data to a txt file if required.

3. The **Plane** tab shows the illustration for the results.

Direction Point

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:

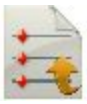
North is the northing coordinate of the point.
East is the easting coordinate of the point.
Elev is the height of the point.
From Point is the name of the known point.
Azimuth from the known point to the unknown point.

3. The **Plan** tab shows the illustration for the results.

Polyline Report

To calculate the inverse data for a known linework:

1. In the **Input** tab:
   - Choose either **Polyline** or **Code** for selecting the linework.
     - *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's 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 from the list.
   - The **Start** station and the horizontal plan of the selected linework will be shown.
   - Click the **Calc** button  to calculate the inverse.
2. In the **Results** tab:
   - Observe the results of the calculation:
     Polyline: the name of the linework.
     Num Segments: the number of segments in the linework.
     Horz Length: the total horizontal length of the linework.
     Start Sta: the starting station of the linework.
     End Sta: the ending station of the linework.
     Segment: information about each segment in the linework.
     Type: the type of the segment.
     Horz Length: the horizontal length of the segment.
     Azimuth: the azimuth of the segment.
     Start Sta: the starting station of the segment.
     End Sta: the ending station of the segment.
     Height: the increment of the height.
     Grade: the increment of the height in percentage terms.

- Click  to save the data to a txt file if required.

3. The **Plane** tab shows the illustration for the results.

Calculate Arcs

Click an icon to calculate the task:

Arc

Calculates the full set of curve parameters, given one length and one radius parameter.

3Pt Arc

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

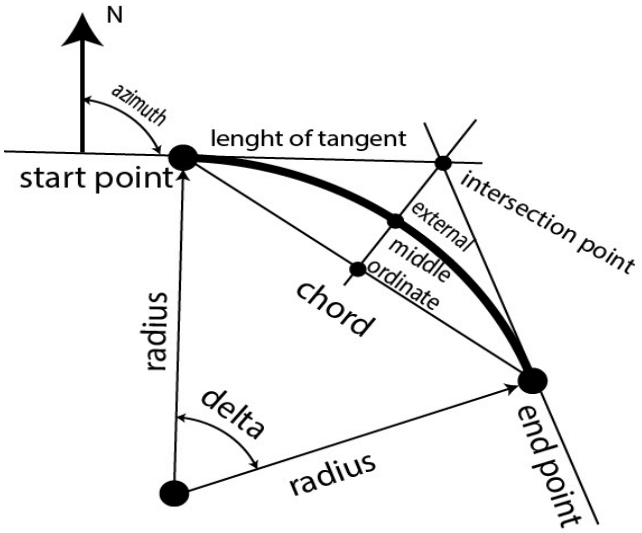
Computes the starting, ending and center points of a curve, given the point of intersection, the radius, and the azimuths from the PI point to the PC and PT points respectively.

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.

Arc

To calculate the full set of arc 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 parameters 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 beween 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 **Plane** tab shows the illustration for the results.

Three-Point Arc

To calculate the arc 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**Plane** tab shows the illustration for the results.

PI & Tangents

To calculate the arc 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 **Plane** tab shows the illustration for the results.

Radius & Points

To calculate the parameters of a arc 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 **Plane** tab shows the illustration for the results.

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 **Plane** 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 linework.

Arc Offset
Defines a curve (section of an arc) used for computing point locations in relation to the curve.

Poliline Offset
Defines a polyline used for computing point locations in relation to the linework.

Multiple Points Offset
Calculates point locations relative to the selected points.

Line Offset

The line from a start point 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 **Direction** 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. Click **Next** to define stationing for offsets and calculate points. **More...**

Along & Offsets

To define stationing, which is used to compute the points locations in relation to lines, curves and alignments.

- If you need to calculate the coordinates of one offset point from the line, select **Along** to set starting station. The two arrow buttons allow you to decrease or increase the station by the interval specified in the station **Interval** field.
  1. Set the station **Interval**.
  2. Enter the **Right Offset** or **Left Offset** of the computed point with respect to the line at the stations.
  3. 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.
  4. Enter the name of the current computed **COGO Point**. Select the code for this point from the

existing codes and, if needed, click  to set the point's attributes. **More...**

5. Click **Calc** to calculate the offset points.

- If you need to calculate the coordinates of several offset points from the line, select **Start Along** to set starting station. The two arrow buttons allow you to decrease or increase the station by the interval specified in the station **Interval** field.
    1. Set the desired **End Along** 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** field.
    2. Set the station **Interval**.
    3. Enter the **Right Offset** or **Left Offset** of the computed point with respect to the line at the stations.
    4. 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.
    5. Enter the name of the current or starting computed **COGO Point**. Select the code for this point from the existing codes and, if needed, click  to set the point's attributes. **More...**
    6. Click **Calc** to calculate the offset points.
- If you need to calculate the coordinates of the offset point which belong a tangent to the line, select **Skew** to set starting station for the tangent.
    1. Click **Right Skew** or **Left Skew** and enter the desired angle of the tangent.
    2. Enter the offset value in the **Skew offset**.
    3. 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.
    4. Enter the name of the current computed **COGO Point**. Select the code for this point from the existing codes and, if needed, click  to set the point's attributes. **More...**
    5. Click **Calc** to calculate the offset points.

Corners Offset

New points are defined by the offset from the nodes of the existing linework. 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 linework segment, connected to the node.

To calculate point locations in relation to a linework nodes:

1. Choose between two options to define the linework:
    - **Linework**: Select an existing linework from the lineworks list or select the linework in the map  . The plot window will display the defined linework.
    - **Code**: Define a code generated line. When Code is selected, the drop-down list will contain a list of all linework 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.

Arc Offset

To calculate point locations in relation to a arc:

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 linework:

1. Choose between two options to define the linework:
   - **Linework***:* Select an existing linework from the lineworks list or select the linework in the map  . The plot window will display the defined linework.
   - **Code***:* Define a code generated line. When Code is selected, the drop-down list will contain a list of all linework 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 Linework***:* To create an offset linework with respect to the selected linework, click **Next**. **More...**
   - **Offset Points From Linework***:* 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 linework and save them in a point list. **More...**

Offset Linework

To create a new **Offset Linework** 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 Linework

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 points' 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.

Offset Multiple Points

To calculate points at an offset from the current job's points:

1. In the **Points/Point List/Linework/Area** field, select points to which the offset will be applied or select groups of points by selecting point lists, linework or area:
   - For **Points**, in the **Sel Pts** drop-down list, choose an option how to select points in the job:

     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 - points found by their name. **More...**
     By Layer - points on a selected layer. **More...**
     From Plan - points selected graphically from the map **More...**
     From List - points from a list of points. **More...**

   - For **Point List**, click  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  .
   - For **Linework** and **Area**, click  and select an existing linework/area from the map (**more...**) or click  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  .

2. The list and the plot will display the points currently selected.
3. If required, move points up and down the list to change the order.
4. To obtain information about a single point highlighted in the list, use the Info icon  .
5. Click  to remove the highlighted point from the list.
6. Click **Next** to set offsets. **More...**

Offset Points

Enter offsets and details for the calculated points:

1. Select a **Method** of the three available to enter appropriate offset data as required:
   - **North, East, Ht**: The northing, easting and height offsets from the points. The labels can change depending on the job settings.
   - **Az, HD, VD**: The azimuth, horizontal and vertical distances from the points.
   - **Az, Slope, Zenith**: The azimuth, slope distance and zenith angle at the points.
2. 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...**
3. Click **Calc** to calculate the offset points.

 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 **Plan** tab shows the illustration for the results.

Triangle

To calculate a triangle:

1. In the **Input** tab:
   - Select a set of three parameters between **Side-Side-Side / Side-Angle-Side / Side-Side-Angle / Angle-Angle-Side / Angle-Side-Angle** and enter them in the corresponding fields.
   - Press **Calc** to calculate the area and other parameters of the triangle.
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 **Plane** 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.

Plane

The Plane icon on the Home screen opens the main Plane. The main Plane displays the plane of the current job. To move the plane display, hold down and slide the stylus on the screen. The Plane maintains the scale after changing the status of the main plane.

The basic commands of the main Plane are available through:

- The plane 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...**

Plane 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 high-light the object(s) you want.

Zoom All
Displays all objects in the plane.

Center to Point
Selects a point for centering the plot.

Layers
Opens the **Layers** dialog.

Plane Properties
Displays the plane properties. The **Plane Properties** are also available from every other pop-up menu.

3D View
Displays 3D view of the plane. 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 plane.

   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 plane 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 plane 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 plane, click the first polyline/arc.
4. On the plane, 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 plane.
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 plane.
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.

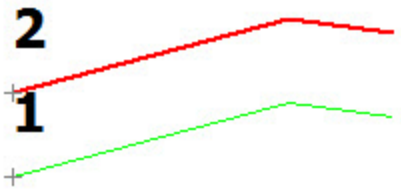
End point snap mode

Use this mode to create:

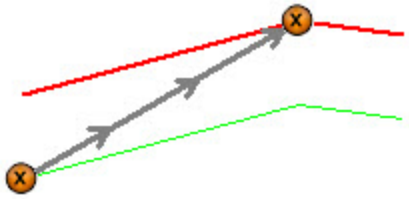
- an end point for the selected segment of the polyline,
- a polyline segment between two end points,
- a closed area with the end points of the selected segment,
- a best fit arc with the end point of the selected segment,
- a best fit line with the end point of the selected segment.

To use this mode:

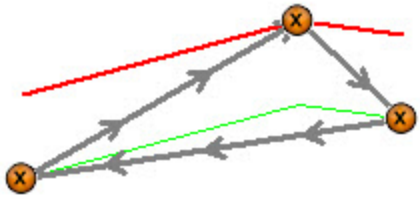
1. Select the drawing tool: **Point**, **Polyline**, **Area** , **Best fit arc**, or **Best fit line** in the **Drawing Toolbar**.
2. On the snap toolbar, click  .
3. Click on a segment to create the end point on the nearest side.
   If the **Point** button is selected, the software creates an end point 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:

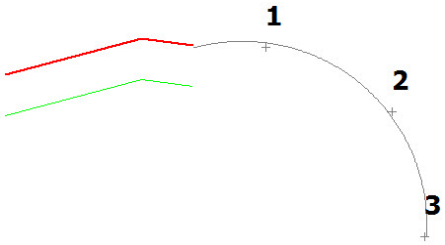


If the **Best fit arc** button is selected, the software creates:

- an end point for the segment that has been clicked

  and after deselecting 

- a new arc as the best fit through the points selected:

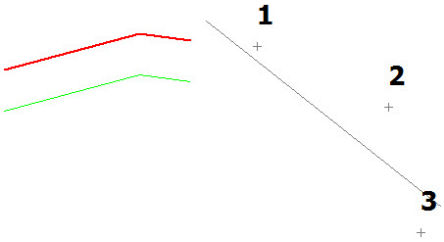


If the **Best fit line** button is selected, the software creates:

- 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:



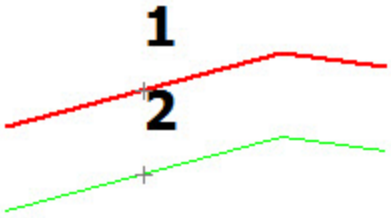
Mid-point snap mode

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:

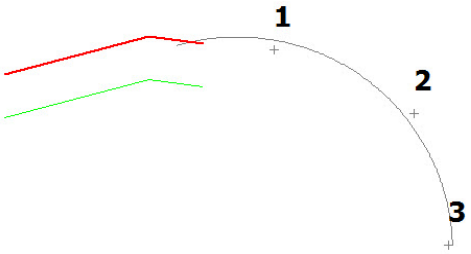
If the **Area** button is selected, the software creates an closure area using the mid points of the segments that have been selected:

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:

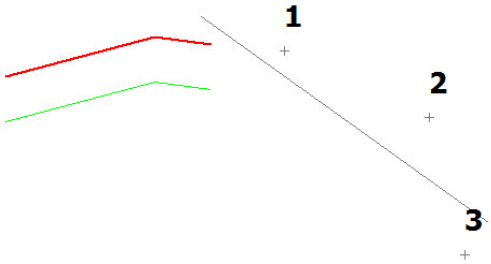


If the **Best fit line** button is selected, the software creates:

- 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:



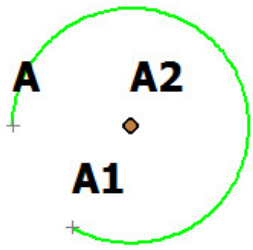
Circle center snap mode

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:

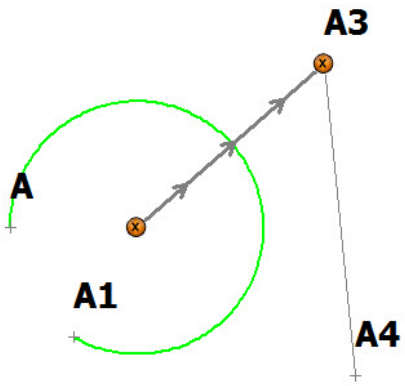
1. Select the drawing tool: **Point**, **Polyline**, **Area**, **Best fit arc**, or **Best fit line** in the **Drawing Toolbar**.
2. On the snap toolbar, click .
3. Click on an arc or circle to create the center point.
   If the **Point** button is selected, the software creates an center point for the 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:

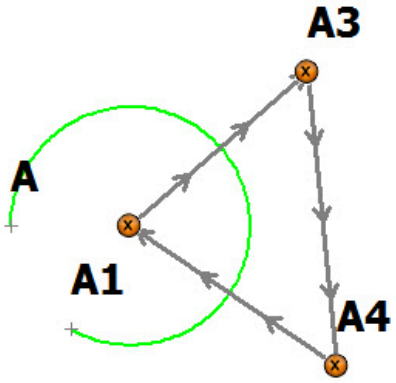


If the **Area** button is selected, the software creates:

- 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:

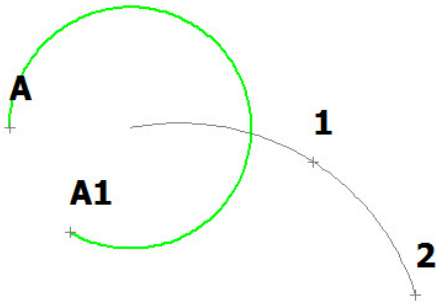


If the **Best fit arc** button is selected, the software creates:

- 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:

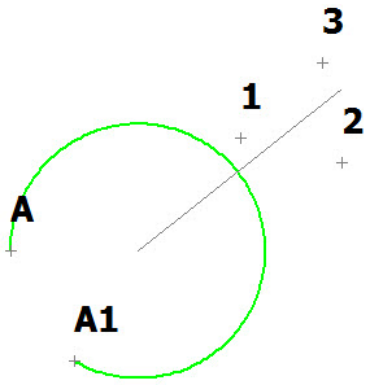


If the **Best fit line** button is selected, the software creates:

- 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:



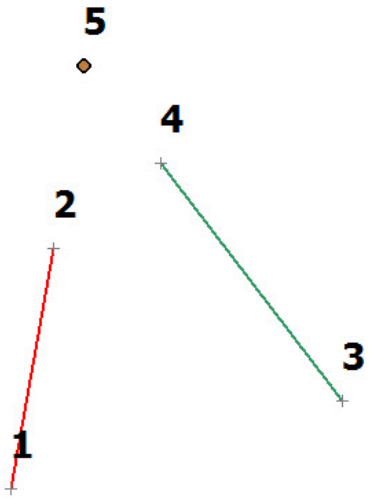
Line intersection snap mode

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:

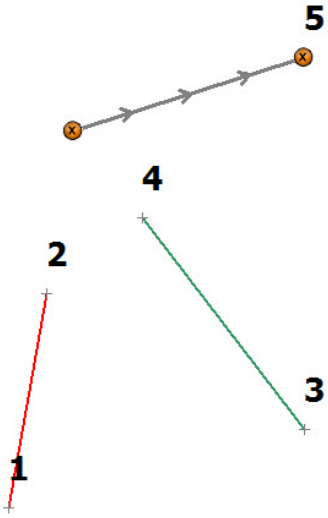
1. Select the drawing tool: **Point**, **Polyline**, **Area**, **Best fit arc**, or **Best fit line** in the **Drawing Toolbar**.
2. On the snap toolbar, click  .
3. Click a polyline/arc and the intersecting polyline/area to create intersecting point:
   If the **Point** button is selected, the software 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:

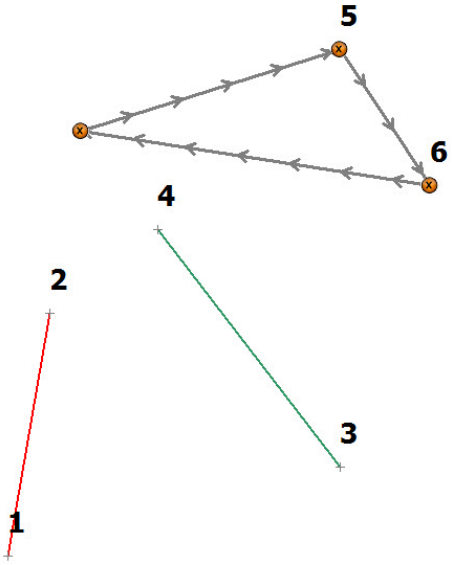


   If the **Area** button is selected, the software creates:

- 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:

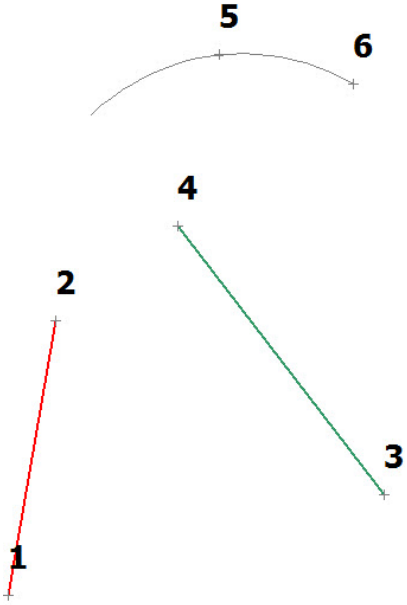


If the **Best fit arc** button is selected, the software creates:

- 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:

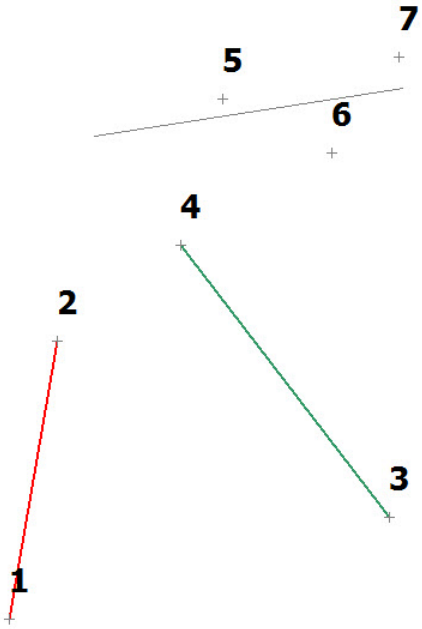


If the **Best fit line** button is selected, the software creates:

- 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.

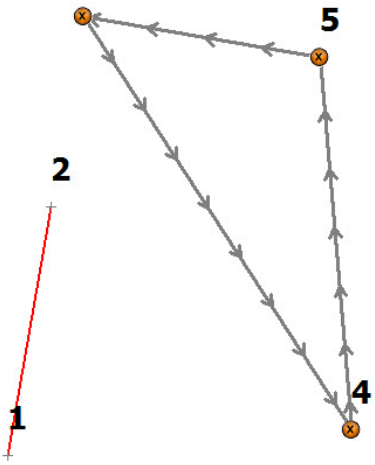
To use this mode:

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 poly-

line 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

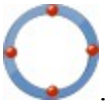
Use this mode to create:

- an point at the intersection of the circle quadrant axe with its circumference,
- a polyline segment between two (and more) intersection points,

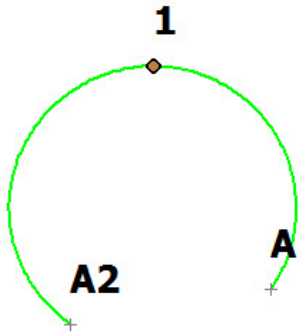
- a closure area from the intersection points,
- a best fit arc with the intersection point,
- a best fit line with the intersection point.

Note: The nearest to the click point intersection will be used.

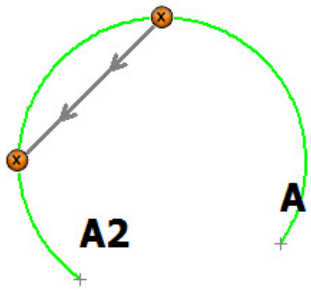
To use this mode:

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:

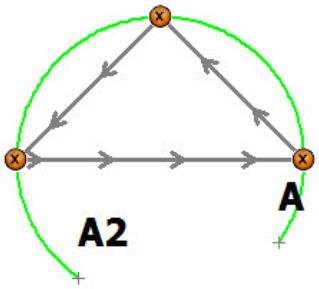
   If the **Point** button is selected, the software creates a point at the intersection of the circle quadrant axes with its selected circumference:



   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:

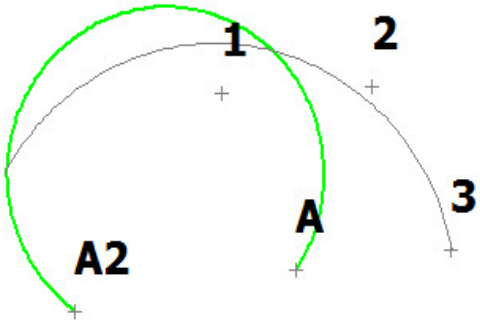


If the **Best fit arc** button is selected, the software creates:

- 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:

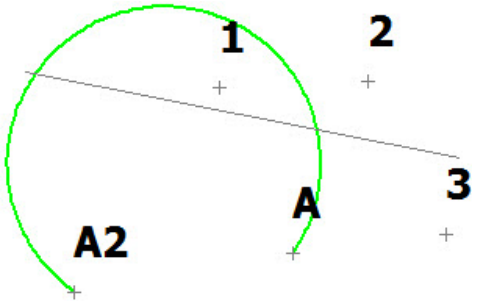


If the **Best fit line** button is selected, the software creates:

- 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:



Plane Pop-Up Menus

To open a pop-up menu hold down the stylus on any place in the plane. The menu options vary depending on the objects selected.

1. When there in no objects selected, the pop-up menu allows you to:
    - Create a point at the place of menu call.
2. When an object is selected, the pop-up menu contains commands which virtually duplicate the number of commands found in MAGNET Field Layout and allow you to:
    - Edit job data
    - Calculate an appropriate COGO task.
    - **Stake** desired objects.
3. If you hold down the stylus on multiple objects, you obtain the **Objects near selected point** dialog that allows you to select individual objects.

Plane Properties

The Plane Properties dialog contains four tabs:

- **General tab**.
- **3D tab**.
- **Drawings tab**.

General tab

In this tab you can configure displaying of the following objects on the plane, 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 Plane the program will automatically load the Bing Maps for the current job objects from the Internet.

When the **Current Position in Main Plan** check box is selected the program automatically snaps back to the center if it moves off the edge of the plane.

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 Plane.

- 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.

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 Plane, select /unselect its checkbox.

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**

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. In **Hybrid Positioning mode**, you can directly toggle between GPS+ and Robotic configurations having both GPS+ receiver and optical robot connected to the MAGNET Field Layout.

- 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. If options of the GPS receiver expired, you will be prompted to check OAF.
- Clicking the **Disconnect** button closes the connection with the current device to connect with another if required.

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.
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. For a MAGNET Relay configuration, the rover starts auto connecting to Enterprise Relay only after the receiver is connected and Enterprise account is connected.

- 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.

Two Peg Test for Level

The Two peg test dialog guides you through a series of measurements to help determine any error in the line of sight through the level.

1. Center a Digital Level (DL) between the two points to measure and take a measurement (by clicking the Measuring button) to point 1 and point 2.
2. Move the DL for the point 2 (approximately 3 meters from the point) and take the shots to the point 1 (by clicking the Measuring button). After that MAGNET Field automatically calculates and displays the true value for the point 2.
3. To compare the true and the measured value, reverse the staff located at point 2. Collimate the staff on the point and read the value manually. If necessary, remove the eyepiece cover to expose the cross-hair adjusting screw and shift the horizontal cross-hair line up or down (as required) until the line is coincided with the reticle reading value. After adjustment the DL, perform the step 1 and step 2.
4. To get the relative error of the DL, again reverse the staff located at point 2 and take a measurement to the point.
5. The results of the Two Peg testing is displayed on the **separate screen**.

Two Peg Test Results

The results of the Two Peg testing appear on this dialog after all measurements are taken. The dialog displays all the shots taken and the computed error.

Level Run

Enter information needed to create a Level Run.

1. Enter a **Name** for the Level Run.
2. In the **Note** field, enter a description for the Level Run.
3. Click **Next** to start leveling.

Level Run

This dialog displays information regarding the Digital Level Run in progress.

DL tab

This tab allows you to perform measurements and record them into a fieldbook table.

- The items to be displayed and the order of items to be displayed can be selected by clicking Display Settings on the pop-up menu  .
- The field with the icon  shows the name of the current point. You can select the point from the map, list or enter the name.
- Click **BS** to take the Backsight measurement to the current point. If you select a new point, the **Add Point** dialog is opened. Enter the coordinates of the point in the dialog and click  to take the measurement to the point. In this tab, **BS** measurement has  symbol.
- Click on **SS** to take a Sideshot measurement. In this tab, **SS** measurement has  symbol.
- Click on **FS** to take a Foresight measurement . In this tab, **FS** measurement has  symbol.
- The SumBS-SumFS shows the difference between the sums of BS and FS distances.

Data

This tab lists all information related to the current measurement for the level run.

Shots

Guides you through the sets needed to be taken for the measurement.

Click **Meas** to take a shot.

Pop-up Menu

The Level Run pop-up menu contains the following options:

- **Stake Point**
- **Stake Point List**
- **Stake Elevation**
- **Vertical Offset**
- **Display Settings**
- **Inverse**

Display Setting

Select the columns to be displayed for a level run from this dialog. The order of the columns to be displayed can be changed by selecting the display column and clicking the up and down arrow.

Vertical Offset

Enter the **Vertical Offset** for the measurement.

Manual Level Entry

This dialog allows you to enter level observation data manually. They can be:

- The height of the **Top Wire** (if using a 3 Wire instrument).
- The height of the **Middle Wire**.
- The height of the **Bottom Wire** (if using a 3 Wire instrument).
- The horizontal **Distance** measurement between the level and the rod.

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 (Total Station) survey

Setup GPS folder

Click an icon to perform the task:

Status
Displays information about the current position of the GNSS receiver, RTK status, and the satellite constellation.

Localization
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.

Simulator
Sets initial WGS84 position for GPS simulation configured. The icon is enabled, when the Simulation Mode is selected in the **Config: Receiver Make** dialog.

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 tab**
- **System tab**
- **Multi Base Status tab**
- **Scatter Plots tab**
- **SVs tab**

Position tab

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 tab**
- **Multi Base Status tab**
- **Scatter Plots tab**
- **SVs tab**

The icon  opens a pop-up menu which varies depending on the configuration type used. **More...**

System tab

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 Layout in this mode.

Other pages in the Status dialog:

- **Position tab**
- **Multi Base Status tab**
- **Scatter Plots tab**
- **SVs tab**

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

---

Multi Base Status tab

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 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 tab**
- **System tab**
- **Scatter Plots tab**
- **SVs tab**

Scatter Plots tab

The Scatter 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  . See Properties for Horizontal Plot and Properties for Vertical Plot.

Other pages in the Status dialog:

- **Position tab**
- **System tab**
- **Multi Base Status tab**
- **SVs tab**

Properties of Horizontal Plot

In this dialog you can show or hide the local coordinate axes (**Show Grid** checkbox) and activate automatically scales the horizontal scatter plot to fit into the dialog (**Auto Zoom** checkbox).

Properties of Vertical Plot

In this dialog you can specify a duration in seconds for the time axis(**Time Window** field).

SVs tab

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 tab**
- **System tab**
- **Multi Base Status tab**
- **Scatter Plots tab**

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 Beacon is available, when you use the beacon as the corrections source for DGPS Real time.
Reset RTK or **Reset DGPS** - the command that reinitializes the receiver.
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.

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).

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.

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

9. To change Mission Planning Setting, click  .

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.
4. Click  to confirm the settings and to calculate and show the position of satellites in the first ten minutes of the specified time on the *Sky Plot* of the Mission Planning dialog. In the combo box you can select the showing **Satellite List**, **Satellite Number**, **PDOP**.

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.

Localization

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 predefined or created by the user grids and datums. In this case MAGNET Field Layout 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 Layout 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 Layout . 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 Layout , 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 Layout, 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 Layout 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 Layout 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 Layout will use this geoid model for calculating orthometric height (elevation).

The current version of MAGNET Field Layout 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.

Simulator

In the **Simulation Setup** dialog you can 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 Layout runs.

Moving speed and direction can be changed in the main map or in the Topo dialog using arrows.

Setup Optical folder

Click an icon to perform the task:

Occupy Point
Sets up a Total Station survey with a reference direction.

Reference Points
Computes the coordinates of an occupation point, where the instrument is set up, using measurements to two (or more) points with known coordinates. When you work in the **Hybrid Positioning mode**, on one known point GPS and TS measurements will be performed.

Reference Elevation
Computes the elevation of an occupation point, where the instrument is set up, using measurements to two (or more) points with known elevations.

Reference Line
Computes the coordinates of an occupation point, where the instrument is set up, using measurements to two design points (or offsets from them). You consider the measurements from the instrument as perfect.

Localization
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.

Remote Ctrl
Transmits commands from the controller to the Robotic total station only.

Occupy Point

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 reference points setup. 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 Offset**from 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...**

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.
- *Length*: opens the **Length** 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 Layout 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.

Reference Points

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.

Reference Points Settings

In the dialog you can select:

1. The target type in the **Measure Reference Points with** field:
   - If you use a reflective prism, click the **Prism** button.

     Note: In the Prism mode you can select HA/VA measurement type and prism is not required in that case.

   - If you do not use a reflective prism and would like to survey in Non-Prism mode, click the **Laser** button.
2. The calculation method of the occupation point elevation in the **Instrument vertical location** field:
   - If you need to obtain the occupation point coordinates in the horizontal plane only, click
     - the **Use 0** button. In this case, the software sets the zero value for the occupation point elevation.

       or

     - the **First Point** button. In this case, the software sets the first point elevation for the occupation point elevation.
   - If you need to obtain the occupation point coordinates in the horizontal and vertical planes, click the **Average** button. Enter the height of the instrument (HI). In this case the vertical component of all referenced points will be used for calculation the occupation point elevation.

Reference Points 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...**

Reference Points 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.

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. You can see the pairs point are used for localization in the **Localization Result** dialog.
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.

Results Screen

The screen contains the data collected during the 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.
- **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:

- **Add**: to continue measure an other control point.
- **Re-Meas**: to repeat the measurement to the previous point
- **Remove**: to delete any selected measurement.
- **Accept**: to calculate (or recalculate, when you add a measurement from the given TS occupation point to the next control points) the occupation point coordinates and finish the resection procedure.

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.
- *Inverse*: 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.
- To apply the calculated scale factor value for the TS occupation point, select the **Use Calculated scale factor** check box.

Reference Elevation

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 **Occupy**, 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.

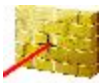
Click the icon  to toggle between the status bar and the tool bar.

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 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.

Plan

The **Plan** tab shows the points in the graphic mode. The buttons on the right duplicate the controls on the first tab.

Survey folder

Survey types depend upon the current job configuration.

Follow a link to learn more:

Survey GPS

Survey Optical (Total Station)

As-built

The dialog enables you to conduct a stationary survey and can run in two views: **Normal** and **Map**.

Click  to open the pop-up menu of additional options. **More...**

Topo Normal View

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. Observe **Status** information about the equipment you use in the job.
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 to 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. The display of the current coordinates depends on the selected coordinate system
7. 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.

Plan View

The Plan View shows the stored points graphically. All the survey process can be done through this view as well. It contains the information similar to the Normal View.

Click on any data field to select the value to be displayed. **More...**

Select Value to display

Select the required item from the list and click  to display it's value during the survey.

Survey pop-up menu

The pop-up menu includes additional options which can be useful during the survey. The menu content depends on the job configuration. The items with small arrows have submenus. Clicking on such a menu item opens a sub-set of items.

- **View** - click and select either type of display of the survey progress:
  Normal
  Plan view on the map
- **Measure** - click and select the offset type for measurements.
- **Setup** - click to expand the menu:
  Simulator - select to move the position on the map with the controller's joystick.
  PTL Mode
  Grid Lines
  Reset RTK - the command that reinitializes the receiver. An error may occur during the
- **Status**
- **Edit Points**
- **Lenght**
- **Quick Codes** - click to expand the menu:
  Show Quick Codes - select to show code boxes on the Map to take measurements with these codes.
  Edit Quick Codes
- **Chats** - click and select **Create New** to start a new chat.
- **Raw Data** - click to expand the menu:
  Add Raw Note
  View Last Point
  Edit

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.

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.

Last Point Data

The dialog contains the information about the last point that has been logged since this dialog was opened.

Click  to save the data to a txt file if required.

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.

Measure

This 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.

As-built

The optical topo survey can be conducted as collection of:

- **Sideshot** points
- **Traverse** points

Data collection is available in **Normal** (dialog) View and **Plan** View.

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

The pop-up menu includes additional options which can be useful during the survey. The menu content depends on the job configuration. The items with small arrows have submenus. Clicking on such a menu item opens a subset of items.

- **View** - click and select either type of display of the survey progress:
  Normal
  Plan view on the map
- **Measure** - click and select:
  Sideshot - to measure separate points.
  Traverse - to measure traverse points.
  Offset type for measurements. **More...**
- **Setup** - click to expand the menu:
  Backsight
  PTL Mode
  Display Coords - select to display coordinates of the point in the Normal View when measured.
  Grid Lines
- **Robotic** - click to expand the menu:
  Remote Settings
  Tilt
- **Image**: Mirror Image, Record Wide Image, Record Tele Image (telescope view with the crosshair), Image as SXGA (standard of resolution).
- **Edit Points**
- **Inverse**
- **Quick Codes** - click to expand the menu:
  Show Quick Codes - select to show code boxes on the Plan 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.
- **Chats** - click and select **Create New** to start a new chat.
- **Raw Data** - click to expand the menu:
  Add Raw Note
  View Last Point
  Edit

Sideshot-Direct

The *As -built* 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. Take measurements. **More...**

Traverse

If you select Traverse survey, the occupation point will automatically change to the previous foresight point and the backsight point will automatically change to the previous occupation point.The **Backsight** dialog will automatically update for the next occupation point.

Offsets

Switch to an appropriate offset task to determine inaccessible points by setting offsets from measured points as required:

Horizontal Angle Offset
Determines a point using the horizontal angle from one point and the distance to another.

Horz-Vert Angle Offset
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.

Two Line Intersection
Determines a point by the intersection of the two lines. Each line is defined by two points or two measurements.

Line and Corner
Determines a point on the corner using one line defined by two points and a horizontal angle measurement.

Line and Offset
Determines a point that is in a distance from a line defined by two points.

Plane and Corner
Determines a point (Corner) by a plane defined by three points and horizontal and vertical angle measurements.

Offsets in MAGNET Field Layout

When surveying with GPS receivers and Total Stations (TS), you may determine points using different types of offsets.

If performing a survey with a GPS receiver, select the GPS device type in the **Connections** dialog.

If performing a survey with a TS instrument, select the Optical device type in the **Connections** dialog.

Offsets for TS Survey

Offset points are available for surveying only for the Sideshot-Direct measurement method. Select this method for the **Topo** survey in the **Mode** dialog (by clicking  in the **Sideshot-Direct** dialog). To select the corresponding offset type, click  in the top left corner and highlight **Measure** in the pop-up menu. You see the following offset types (follow the link to find out more).

Horizontal Angle Offset	**Horizontal Angle Offset**
Horz/Vert Angle Offset	**Horizontal/Vertical Angle Offset**
Distance Offset	**Distance Offset**
Hidden Point	**Hidden Point**
Two Line Intersection	**Two Lines Intersection**
Line and Corner	**Lines and Corner**
Line and Offset	**Line and Offset**
Plane and Corner	**Plane and Corner**

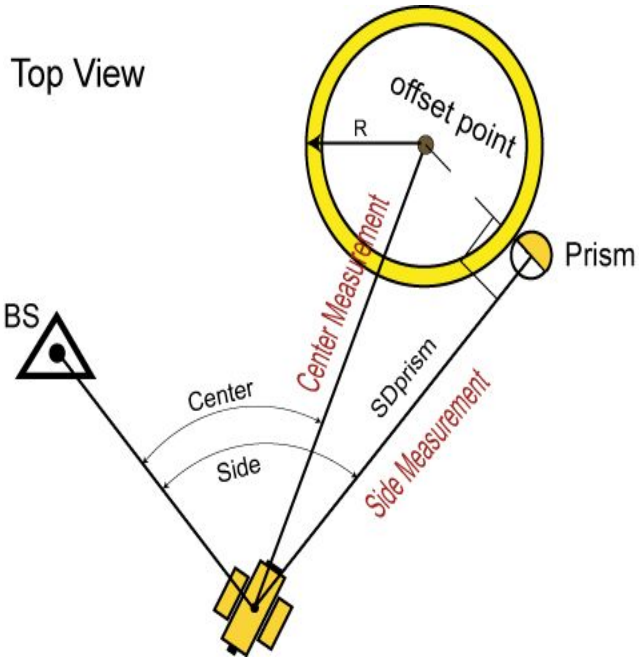
Horizontal Angle Offset

What task does this offset type perform?- Determination of horizontal (2D) coordinates of an inaccessible center of pipe / tree / pillar using distance and angle measurements.

To do this determination:

1. Enter the name of the determined point in the **Point** field of the **Sideshot-Direct** dialog.
2. Click  in the top left corner, highlight **Measure** in the pop-up menu and select Horizontal Angle Offset .
3. The **Horizontal Angle Offset** is displayed.
4. Take two measurements in any order:
   1)  Click the Side button, and measure the distance and horizontal angle to the side of the pipe / tree / pillar. The prism is located on the side of the pipe / tree / pillar.
   2)  Click the Center button, and measure the horizontal angle to the center of the pipe / tree / pillar.

The Horizontal Angle Offset is displayed on the picture:



Note: The offset point lies on the perpendicular to the line Station- Prism at the Prism point.

The **Points** dialog will display the coordinates of this offset (center) point. The vertical measurements to this offset point are not considered, and Hoffset = Hside

---

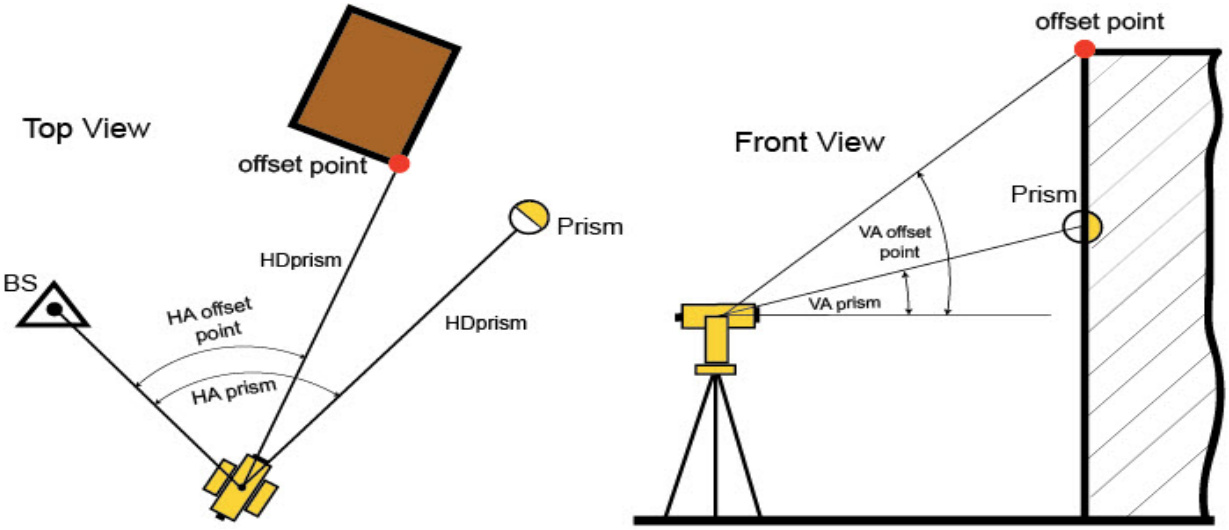
Horizontal/Vertical Angle Offset

What task does this offset type perform?- Determination of the horizontal and vertical (3D) coordinates of an inaccessible point distance and angle measurements.

To do this determination:

1. Enter the name of the determined point in the **Point** field of the **Sideshot-Direct** dialog.
2. Click  in the top left corner, highlight **Measure** in the pop-up menu and select Horz/Vert Angle Offset .
3. The **Horz/Vert Angle Offset** is displayed.
4. Take two measurements in the following order:
   1)  Click the Prism button, and measure the distance and horizontal angle to the prism. The prism is located near the inaccessible point.
   2)  Click the HA/VA button, and measure the horizontal and vertical angle to the inaccessible point.

The Horizontal/Vertical Angle Offset is displayed on the picture:



Note: The offset point lies on the perpendicular to the line Station- Prism at the Prism point.

The **Points** screen will display the coordinates of this offset point.

Distance Offset

What task does this offset type perform?- Determination of the horizontal and vertical (3D) coordinates of an inaccessible point using distance and angle measurements, and additional distance offsets measurements, for example performed with the tape.

To do this determination:

1. Enter the name of the determined point in the **Point** field of the **Sideshot-Direct** dialog.
2. Click  in the top left corner, highlight **Measure** in the pop-up menu and select 

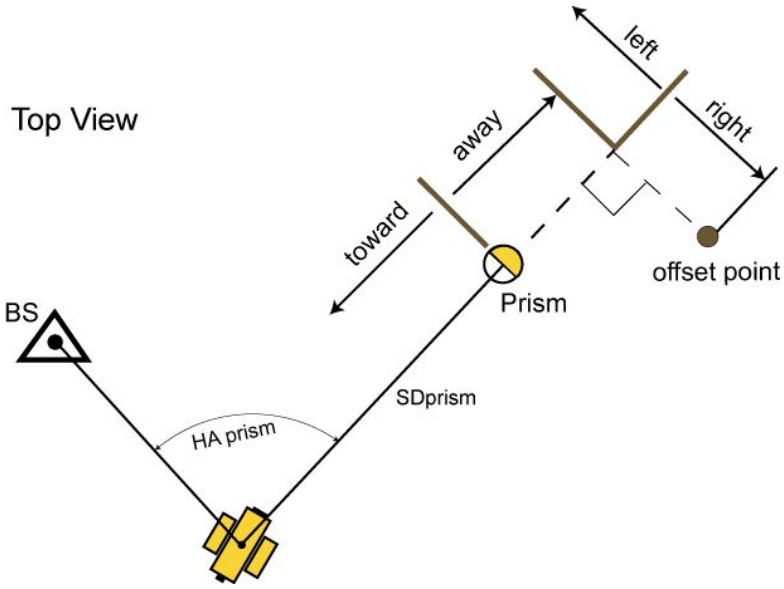
.
3. The **Distance Offset** dialog is displayed.
4. Click 

to take measurements to the Prism.
5. Then measure offsets from the Prism to the determined point and enter their values in the corresponding fields of the Enter Distance Offsets screen. You can define the direction of the given offset in two ways: either click on the button of the current selection or use the minus sign "-" for the current value. For instance, 

and 

directions are the same.

The Distance Offset is displayed on the picture:



Note 1: The offset point lies on the perpendicular to the line Station- Prism.

Note 2: Hoffset_point = Hprism_point + (Up/Down_offset).

The **Points** screen will display the coordinates of the offset point.

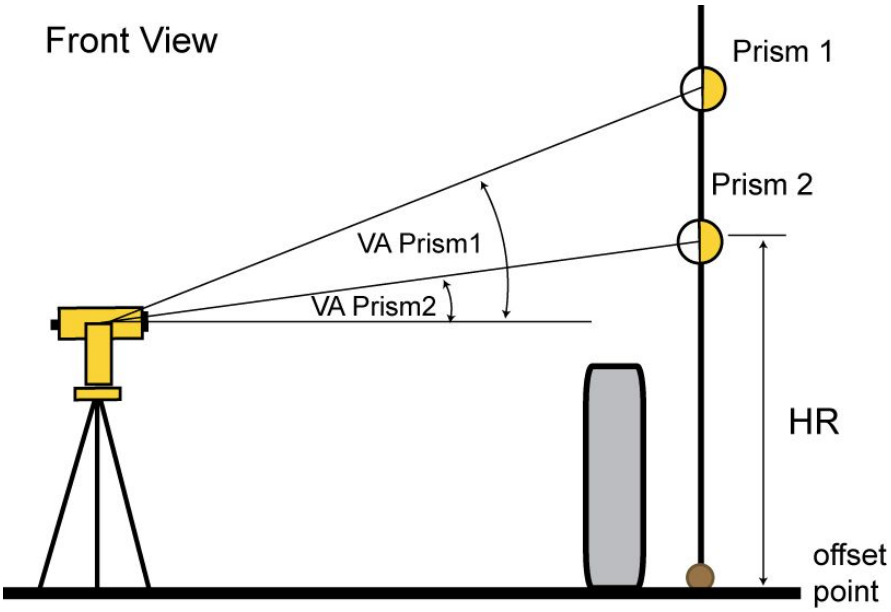
Hidden Point

What task does this offset type perform?- Determination of the horizontal and vertical (3D) coordinates of an inaccessible point using distance and angle measurements to two prisms located on the rod.

To do this determination:

1. Enter the name of the determined point in the **Point** field of the **Sideshot-Direct** dialog.
2. Click  in the top left corner, highlight **Measure** in the pop-up menu and select Hidden Point .
3. The **Hidden Point** dialog is displayed.
4. Enter the distance from the rod's bottom to Prism 2 in the HR field.
5. Take two measurements in any order:
   1)  Click the Prism1 icon and measure the distance, vertical and horizontal angle to the first prism of the rod.
   2)  Click the Prism2 icon and measure the distance, vertical and horizontal angle to the second prism of the rod.

The Hidden Point Offset is displayed on the picture:



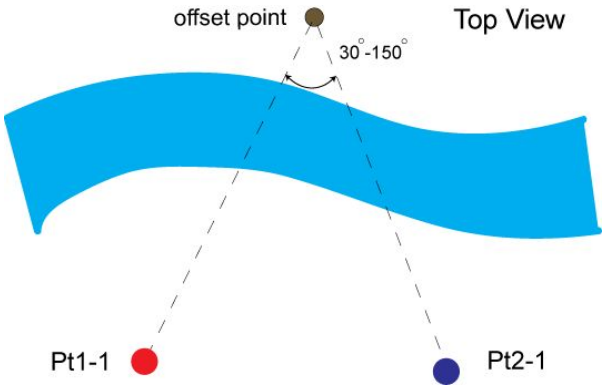
The **Points** screen will display the coordinates of this offset (hidden) point.

Two Lines Intersection

What task does this offset type perform?- Determination of the horizontal and vertical (3D) coordinates of an inaccessible point, as intersection of two auxiliary lines. You create these lines (by two points) and perform distance and angle measurements of the line's points.

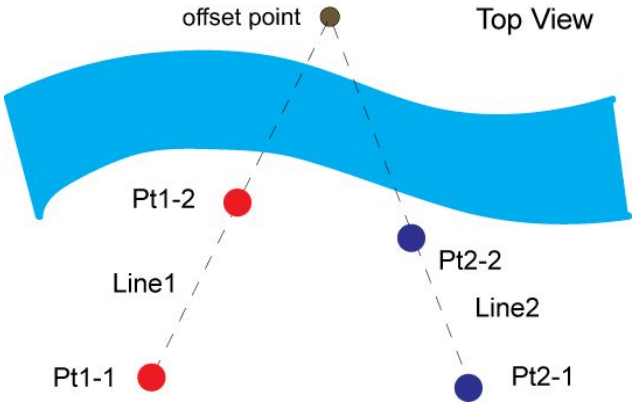
Before performing any measurements, complete the following preparation tasks:

1. Select any two points (Pt1-1 and Pt2-1), from which the desired inaccessible offset point is seen:



When selecting the location of these points, bear in mind that the internal angle formed by the two lines should not be less than 30 degrees and more than 150 degrees.

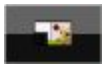
2. Set two more points (Pt1-2 and Pt2-2) along the lines "Pt1-1 - offset point" and "Pt2-1 - offset point", respectively:

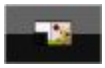


To do this offset determination:

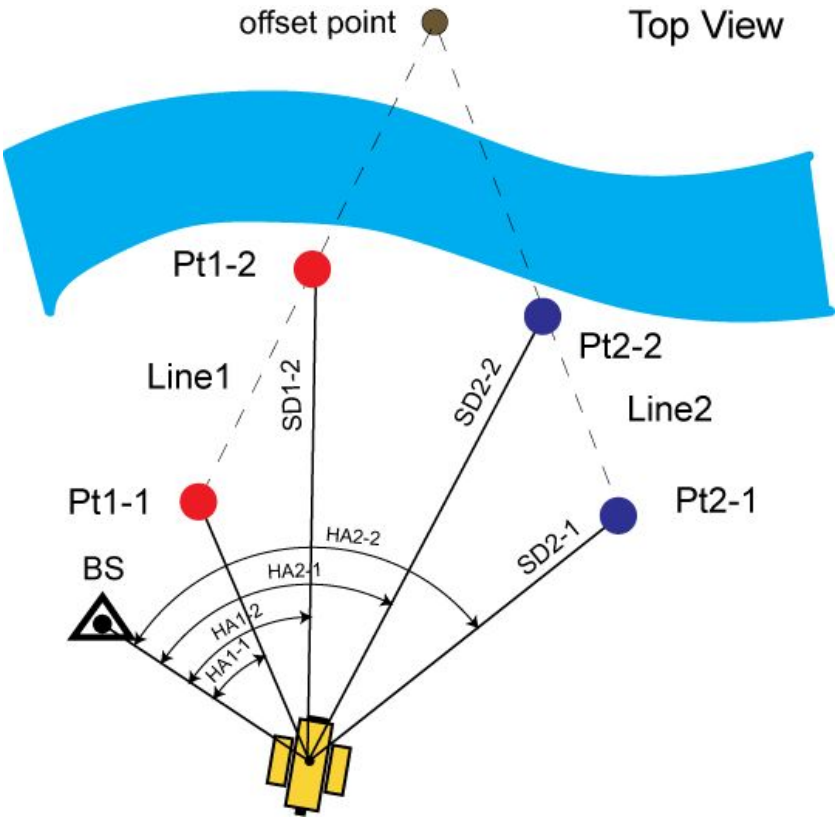
1. Enter the name of the determined point in the **Point** field of the **Sideshot-Direct** dialog.
2. Click  in the top left corner, highlight **Measure** in the pop-up menu and select Two Line Intersection.
3. The **Two Line Intersection** dialog is displayed.
4. Take four measurements in any order:
   1)  Click the Line1Pt1 button and measure the distance, vertical and horizontal angle to the first point of the first line.
   2)  Click the Line1Pt2 button and measure the distance, vertical and horizontal

angle to the second point of the first line.

3)  Click the Line2Pt1 button and measure the distance, vertical and horizontal angle to the first point of the second line.

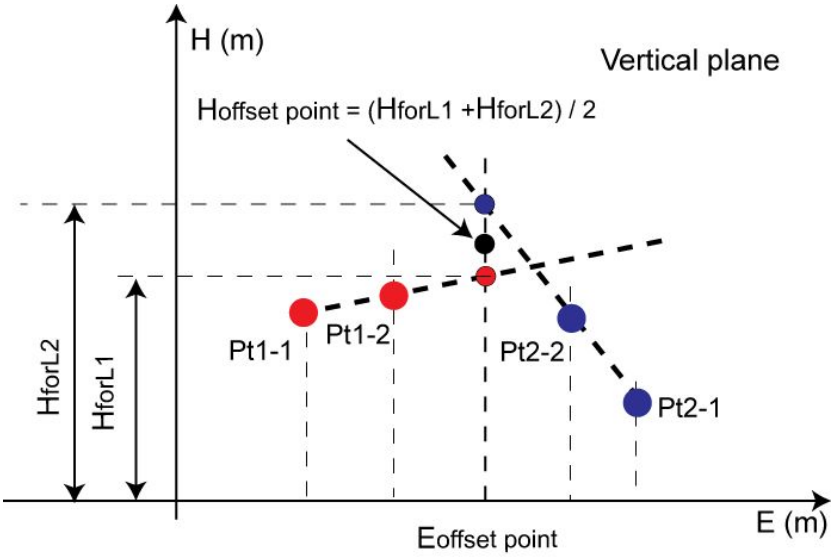
4)  Click the Line2Pt2 icon and measure the distance, vertical and horizontal angle to the second point of the second line.

The Two Lines Intersection Offset is displayed on the picture:



Note 1: The height of the prism for all measured points is unavailable in the **Two Line Intersection** dialog and should be set in the **Backsight** screen.

Note 2: The height of the offset point is calculated as the mean of the heights of both lines for this point:



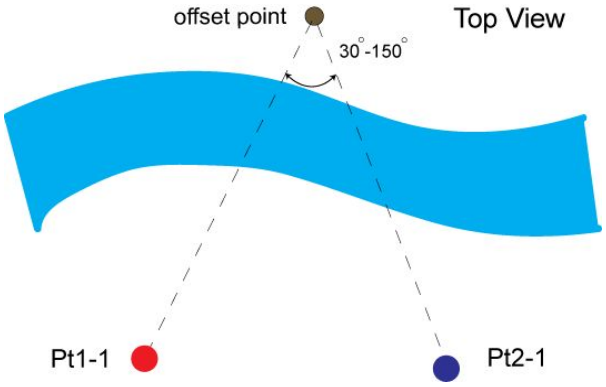
The **Points** screen will display the coordinates of the offset (intersection) point.

Line and Corner

What task does this offset type perform?- Determination of the horizontal and vertical (3D) coordinates of an inaccessible point, as intersection an auxiliary line and a plane. You create a line (by two points) and a vertical plane passing through the offset point. Then you perform distance and angle measurements of the line's points and angle measurements of the offset point.

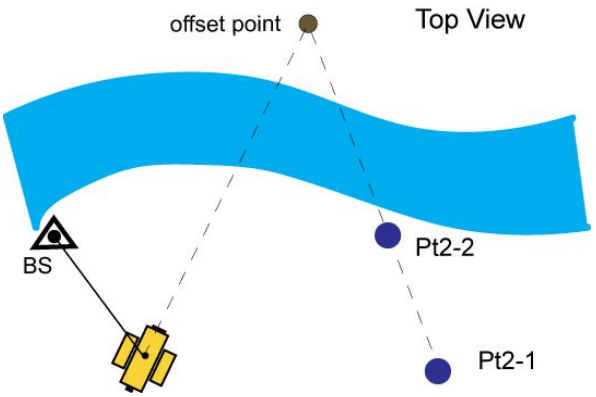
Before any measurements, complete the following preparation tasks:

1. Select any two points (Pt1-1 and Pt2-1), from which the desired inaccessible offset point is seen:

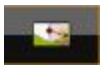


When selecting the location of these points, bear in mind that the internal angle formed by the two lines should not be less than 30 degrees and more than 150 degrees.

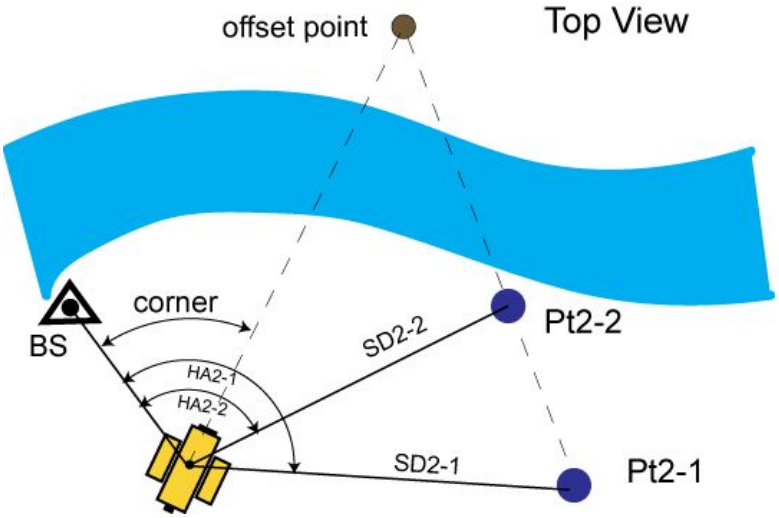
2. Set point Pt2-2 along the line "Pt2-1 - the offset point" and set up the TS at point Pt1-1:



To do this offset determination:

1. Enter the name of the determined point in the **Point** field of the **Sideshot-Direct** dialog.
2. Click  in the top left corner, highlight **Measure** in the pop-up menu and select Line and Corner .
3. The **Line and Corner** dialog is displayed.
4. Take three measurements in any order:
   1)  Click the Line1Pt1 button and measure the distance, vertical and horizontal angle to the first point of the line.
   2)  Click the Line1Pt2 button and measure the distance, vertical and horizontal angle to the second point of the line.
   3)  Click the Corner button and measure the vertical and horizontal angle to the off-set point.

The Line and Corner Offset is displayed on the picture:



Note 1: The height of the prism for all measured points is unavailable in the **Two Line Intersection** dialog and should be set in the **Backsight** screen.

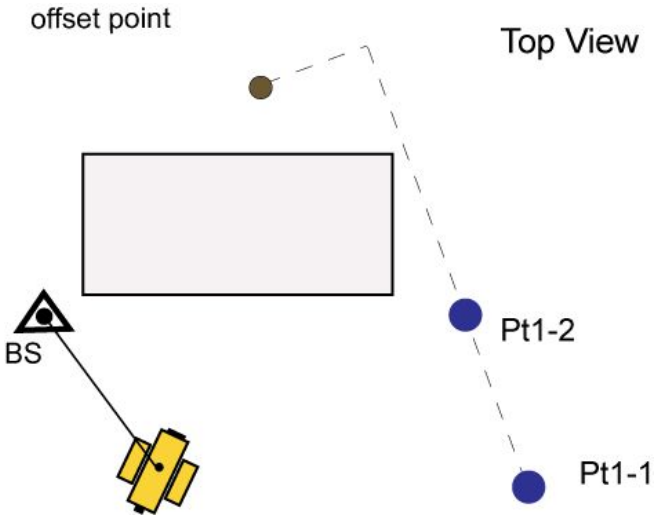
Note 2. The height of the offset point is the height of the line for this offset point (calculated by extrapolation along the “Pt2-1 - Pt2-2” line).

The **Points** screen will display the coordinates of this offset point.

Line and Offset

What task does this offset type perform?- Determination of the horizontal and vertical (3D) coordinates of an inaccessible point, using an auxiliary line and additional distance offsets measurements, for example made by the tape. You create a line (by two points), then perform distance and angle measurements of the line’s points and distance measurements (from this line) to the offset point.

Before any measurements, you have to select any two points (Pt1 and Pt2) near the inaccessible point. The line which is created by these points will be an auxiliary line.



To do this offset determination:

1. Enter the name of the determined point in the **Point** field of the **Sideshot-Direct** dialog.
2. Click  in the top left corner, highlight **Measure** in the pop-up menu and select Line and Offset .
3. The **Line and Offset** dialog is displayed.

4. Take two measurements in any order:

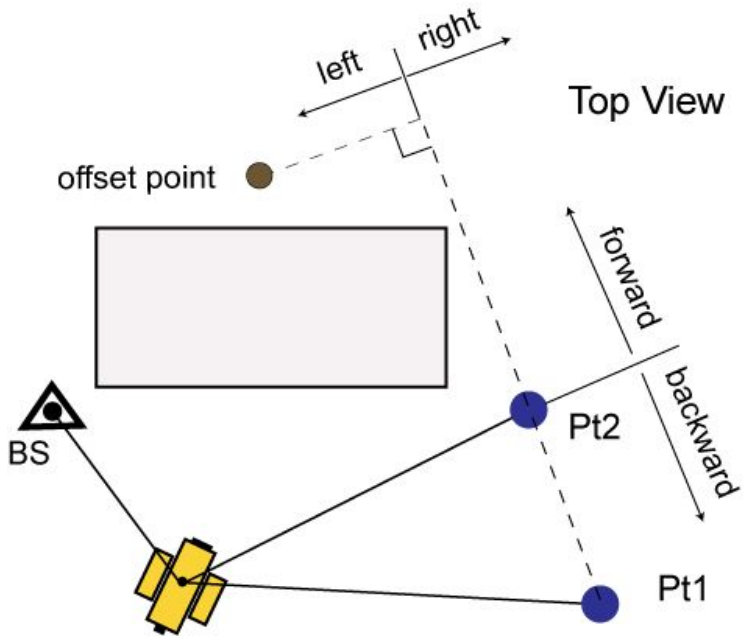
   1)  Click the LinePt1 button and measure the distance, vertical and horizontal angle to the first point of the line.

   2)  Click the LinePt2 button and measure the distance, vertical and horizontal angle to the second point of the line.

5. Measure the offsets from the second point (Pt2) to the determined point and enter their values in the following fields of the Enter Distance Offsets screen. You may define the direction of the given offset in two ways: either click on the button of the current selection or use the minus sign "-" for the current value. For instance, 

and 

directions are the same.
   The directions of the offsets are explained in the figure:



Note 1: The height of the prism for all measured points is unavailable in the **Line and Offset** dialog and should be set in the **Backsight** screen.

Note 2: Offset point lies on the perpendicular to the line "Pt1 - Pt2".

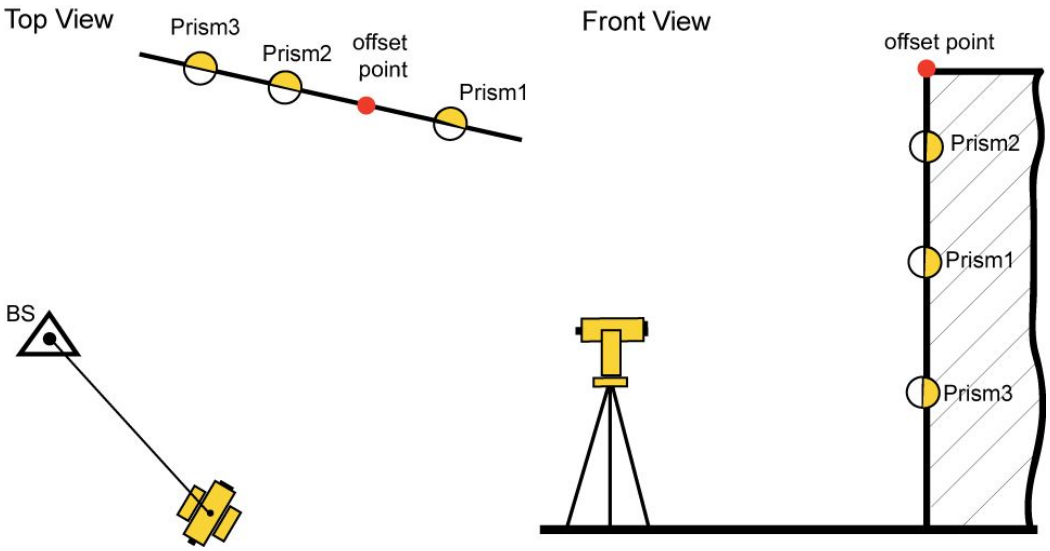
Note 3: 3D coordinates of the offset point are calculated relative to line “Pt1 - Pt2” (calculated by extrapolation along the "Pt1 - Pt2" line).

The **Points** screen will display the coordinates of the offset point.

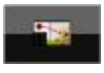
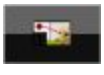
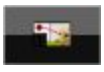
Plane and Corner

What task does this offset type perform?- Determination of the horizontal and vertical (3D) coordinates of an inaccessible point, as intersection of a line of site and a plane defined by three points. You create three auxiliary points, then perform distance and angle measurements to these points and angle measurements to the offset point.

Note that the user creates three auxiliary points in the same plane where the inaccessible point is located. All four points must not be on the same line:

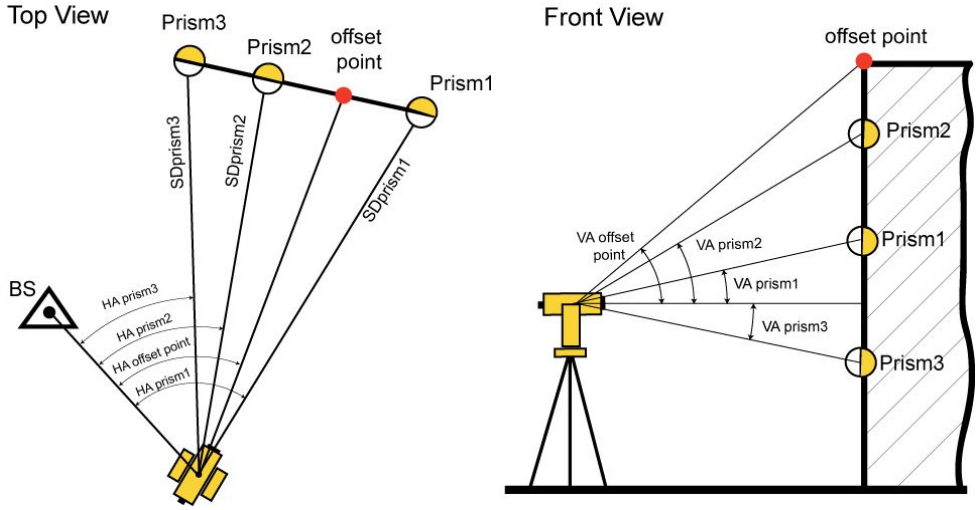


To do this determination:

1. Enter the name of the determined point in the **Point** field of the **Sideshot-Direct** dialog.
2. Click  in the top left corner, highlight **Measure** in the pop-up menu and select Plane and Corner .
3. The **Plane and Corner** dialog is displayed.
4. Take four measurements in any order:
   1)  Click the Prism1 button and measure the distance, vertical and horizontal angle to the first prism of the plane.
   2)  Click the Prism2 button and measure the distance, vertical and horizontal angle to the second prism of the plane.
   3)  Click the Prism3 button and measure the distance, vertical and horizontal angle to the third prism of the plane.

4)  Click the Corner button and measure the vertical and horizontal angle to the offset point.

The Plane and Corner Offset is displayed on the picture:



Note: The height of the prism for all measured points is unavailable in the **Plane and Corner** dialog and should be set in the **Backsight** screen.

The **Points** screen will display the coordinates of the offset point.

Offsets for GPS Survey

The Offsets tab is always presented when you open the **Topo** screen for the GPS survey. The Offsets tab contains up to three offset types. The Offset Laser is available only if a laser distance meter device is used.

Follow the link to find out more:

Offset Line	**Offset Line**
Azimuth & Offsets	**Azimuth and Offsets**
Offset Laser	**Offset Laser**

Offset Line

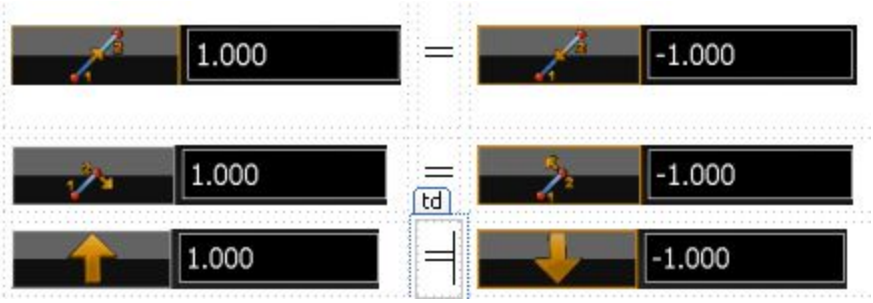
What task does this offset type perform?- Determination of the horizontal and vertical (3D) coordinates of an inaccessible point, using two measured points and additional distance offsets measurements, for example, performed with the tape.

To do this determination:

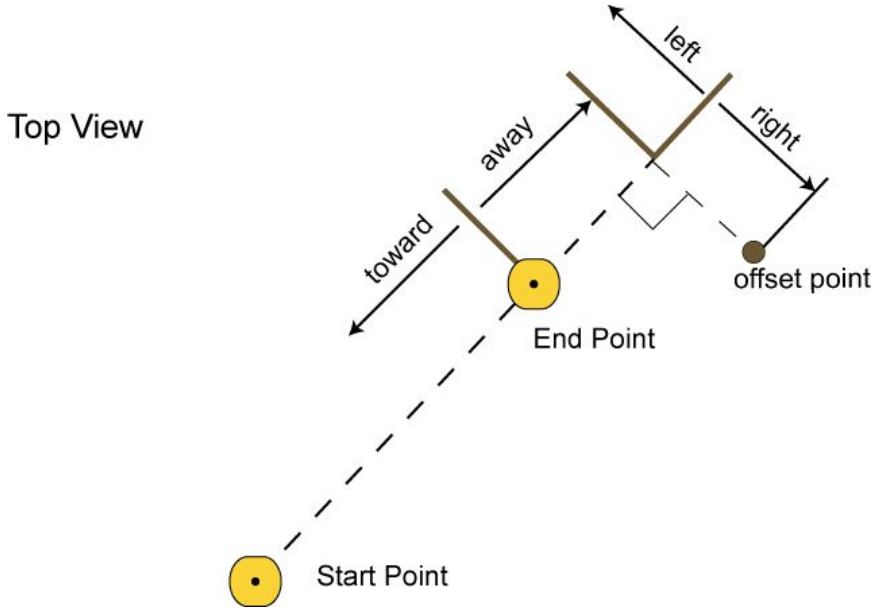
1. Enter the name of the determined point and the rover antenna height in the Point field of the **Topo** dialog.

   Note: Be sure that the current type of position calculation method corresponds to the precision of the determined point.

2. Click  in the top left corner, highlight **Measure** in the pop-up menu and select Offset Laser.
3. The **Offset Line** dialog is displayed.
4. Select the Start and End points from the map (click ) or from the list (click ), or measure them (click ) in the dialog.
5. Measure offsets from the End point to the determined point and enter the values. Use one of two ways to define the direction of the given offset: either click on the button of the current selection or use the minus sign "-" for the current value:



6. The directions of the offsets are explained in the figure:



7. Click  to save the calculated coordinates of the offset point. The **Points** screen will display the coordinates of the offset point.

The offset point lies on the perpendicular to the line "Start Point - End Point".

Hoffset_point = Hpoint2 + (Up/Down_offset)

Azimuth and Offsets

What task does this offset type perform?- Determination of the horizontal and vertical (3D) coordinates of an inaccessible point, using one measured point and distance and angle measurements from this point to the offset point.

To determine a point with the Azimuth & Offsets task:

1. Type the name of determined point and the rover antenna height in the Point field of the **Topo** dialog.

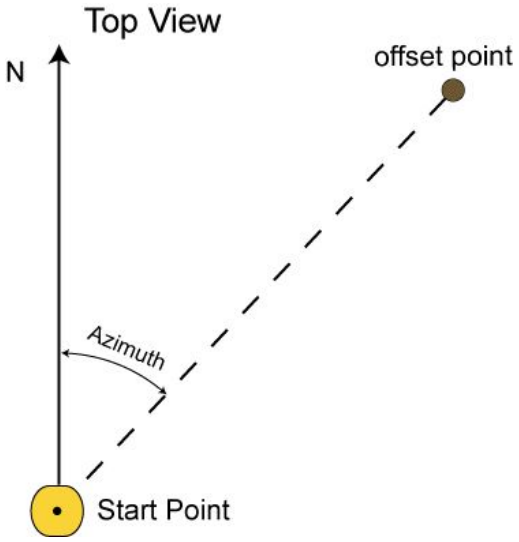
   Note: Be sure that the current type of position calculation method corresponds to the precision of the determined point.

2. Click  in the top left corner, highlight **Measure** in the pop-up menu and select Azimuth & Offsets.
3. The **Azimuth - Distance - Height** dialog is displayed.
4. Select Start Point from the map (click ) or from the list (click ), or measure it (click ) in the Line screen.

5. Define the direction to the offset point in the Azimuth - Distance - Height screen. Use one of two methods to define the offset point in horizontal plane (the hand symbol points out that there is a choice):

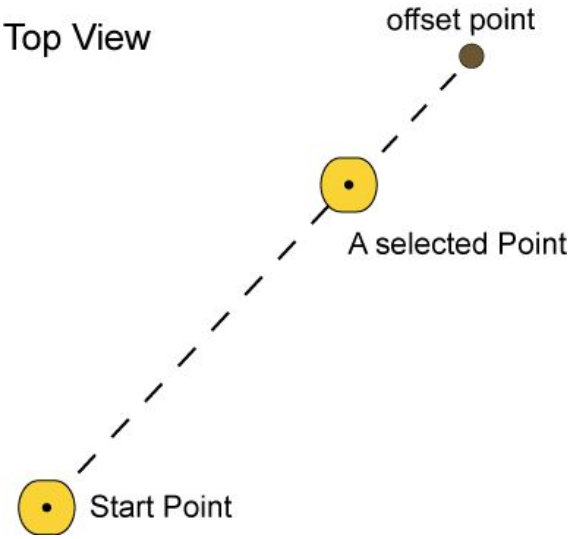
1) Azimuth

The horizontal angle is determined by the azimuth to the offset point.



2) Az to Pt

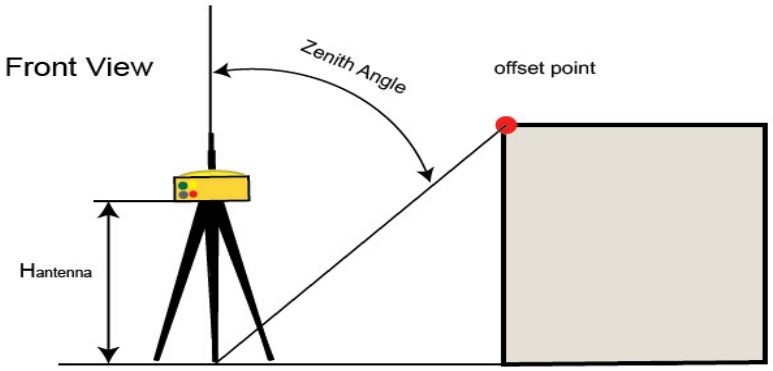
The horizontal angle is determined by the azimuth to a point (selected from the list/map). The offset point lies on the line "Start Point"- "Selected Point".



6. Measure the vertical offset from the Start Point and enter the value in the corresponding field of the Azimuth - Distance - Height screen. Use one of three methods to define the offset point in a vertical plane (the hand symbol points out that there is a choice):

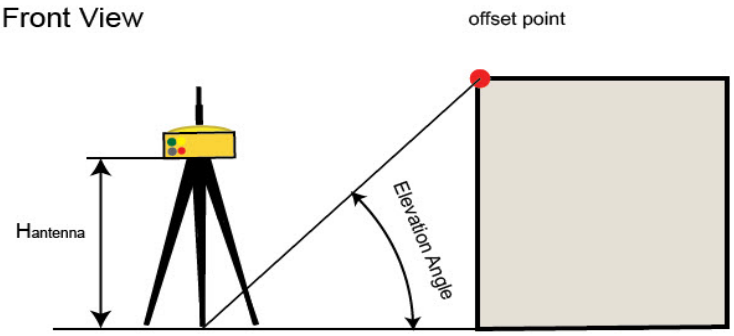
1) Zenith Angle

The zenith angle is measured (from an index with 0 derected towards the zenith).



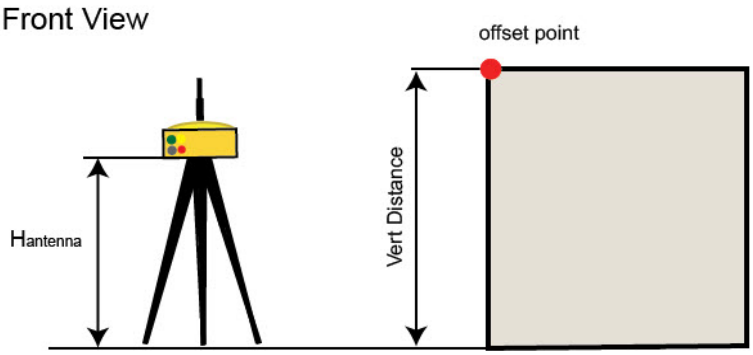
The vertical angle is measured (relative to 0 that coincides with horizontal).

2) Elev Angle



The vertical distance between the offset point and the horizontal passed through the start point.

3) Vert Distance



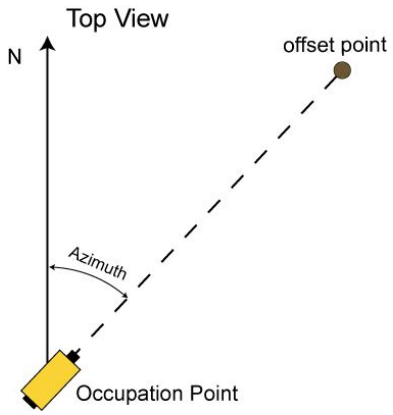
7. Measure the horizontal distance offset from the Start Point and enter the value in the corresponding field of the **Azimuth - Distance - Height** dialog.
8. Click  to calculate and save the coordinates of the offset point. The **Points** screen will display the coordinates of the offset point.

Offset Laser

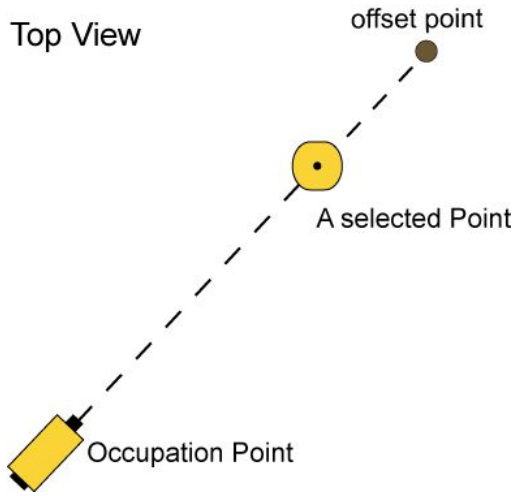
What task does this offset type perform?- Determination of the horizontal and vertical (3D) coordinates of an inaccessible point, using distance and angle measurements to the point from a laser rangefinder. MAGNET Field enables you to use an external laser rangefinder. Before any measurement, set the type of the used rangefinder in job configuration.

To work with an external laser rangefinder:

1. Click the Configure icon  .
2. Click the Survey icon  .
3. Click Peripherals in the **Rover Receiver** screen.
4. Check the External laser option, select which device to connect the laser rangefinder to: Controller or Receiver.
5. Click Parameters in the Peripherals dialog to set communication parameters between the controller and external laser rangefinder.
6. Before starting measurements with the laser rangefinder, make sure that the current coordinate system in the given MAGNET Field job is datum, grid, or localization.
7. Make sure that the job contains the coordinates of the point where the laser rangefinder will be located. This point is called "Occ point" for this offset type.
8. Enter the name of the determined point in the Point field of the **Topo** dialog.
9. Click  in the top left corner, highlight **Measure** in the pop-up menu and select Offset Laser.
10. The **Laser Measure** dialog is displayed.
11. Select the name of the external laser rangefinder from the list in the Bluetooth Devices screen. Click Select in the dialog.
12. Select the Occupation Point from the map (click ) or from the list (click ) in the Config Laser screen.
13. Either enter the azimuth to the offset point (if Azimuth is selected).



14. Or select a point from the list/map (if Az to Pt is selected). In this case the horizontal angle will be determined by the azimuth to the selected point. The offset point will be on the line "Occupation Point"-"Selected Point".



15. Enter the laser rangefinder height in the corresponding field.
16. Sight the laser to the desired point and press the Fire button on the laser rangefinder to measure the slope distance and vertical angle to the point. After measurements are successfully performed, MAGNET Field displays the Store Point screen with the calculated coordinates of the offset point.
17. Click ✓ to save the offset point in the MAGNET Field job. The **Points** screen will display the coordinates of the offset point.

Layout folder

You can perform layout any object using GPS or Optical equipment. Before using GPS receiver for layout task, perform **Localization**.

Click an icon to perform the task:

Points
Stakes design points.

Point List
Stakes points in a point list.

Line by Points
Determines points along desired lines formed by design points.

Arc by Points
Stakes points along a horizontal arc.

Line Offsets
Stakes a line with offsets in the Horizontal and Vertical directions.

Arc Offsets
Stakes a curve (section of an arc) at a specified horizontal and vertical offset from the curve.

3Pt Arc Offsets
Creates a curve to stakeout by selecting three points.

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

Layout 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. **Deviation Report** shows the name of the current layout report if it is set.
6. Click **Layout** to open the **Layout Point** dialog, assisting the stakeout process. **More...**

For the icons available on the dialog see **Icons Description.**

Layout Point List

To stake points of an existing point list:

1. Select the **Name** of existing Point List. You can also enter it manually.
2. The dialog will display a list and the plot of existing points in the selected Point List. You can hide the plot by clicking. The point highlighted in the list will be marked with a yellow circle on the plot.
3. The Up and Down arrows to the left of List of points enables you to move the highlighted point up or down in the order of the points.
4. The arrows symbol switches on/off the keyboard arrow keys to move points inside the list.
5. Select **Reverse Order** to perform a stakeout starting in reverse order from the end of the Point List.
6. 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*.
7. For TS, in **HR**, enter the reflector height.
8. **Deviation Report** shows the name of the current stake report if it is set.
9. Click **Layout** to start the stakeout process for the points. **More...**

For the icons available on the dialog see **Icons Description**.

Layout Line by Points

To layout a line:

1. Define the line:
    - Select the **Start Point** of the line from the map or the list of the job points.
    - Select either the **End Point** of the line in the same way as the start point or select **Azimuth** and enter the azimuth value to define the direction of the line.
2. From the **Height Computation** drop-down list select the type of height computations for the staked point :
    - *Ht of Start Pt* (the height of the start point): the staked point will have the same height as the start point of the line.
    - *Interpolate Ht*: the height of the staked point will be computed through linear interpolation, using the height of the start and end points of the line.
      Note: this option is unavailable when only the direction of the line is set through an azimuth.
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. **Deviation Report** shows the name of the current stake report if it is set.
6. Click **Layout** to open the Stake dialog, assisting the stakeout process. **More...**

For the icons available on the dialog see **Icons Description**.

Layout Arc by Points

To stake a curve (section of an arc):

1. Define the curve. You can manually enter, or select from the list or map the following points:
    - In **PC Point**, enter the Point of Curve, the starting point of the arc.
    - In **PT Point**, enter the Point of Tangent to the curve, the ending point of the arc.
    - Select between **Radius/Chord Angle/Curve Angle** to enter the radius parameter of the curve as required. The plot will show the curve.
2. In **Turn**, select the direction of turn (*Right* or *Left*), relative to the PC Point.
3. Select either *Small* or *Large* from the **Curve** drop-down box to indicate if the smaller or larger part of the arc between the points is used.
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. **Deviation Report** shows the name of the current stake report if it is set.
7. Click **Layout** to start the stakeout process. **More...**

For the icons available on the dialog see **Icons Description**.

Layout Line Offsets

To stake offsets from a line:

1. Define the line:
   - Select the **Start Point** of the line from the map or the list of the job points.
   - Select either the **End Point** of the line in the same way as the start point or select **Azimuth** and enter the azimuth value to define the direction of the line. When the End Point is selected, the **Length** of the line is displayed.
2. From the **Height Computation** drop-down list select the type of height computations for the staked point:
   - *Ht of Start Pt* (the height of the start point): the staked point will have the same height as the start point of the line.
   - *Interpolate Ht*: the height of the staked point will be computed through linear interpolation, using the height of the start and end points of the line.
     Note: this option is unavailable when only the direction of the line is set through an azimuth.
3. Select the **Include Transition Point** checkbox if the transition point should be included.
4. In **Start**, enter the line station (chainage) to begin staking offsets.
5. **Deviation Report** shows the name of the current stake report if it is set.
6. Click **Next** to proceed on the **Station & Offsets** dialog.

For the icons available on the dialog see **Icons Description**.

Station & Offset

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** to be applied in plane.
3. As required, enter the **Up**/**Down** offset to be applied in height.

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. Click **Back** to return to the previous initial dialog if required.
7. Click **Layout** to start the stakeout process for the designated offset locations. **More...**

Layout Arc Offsets

To stake a curve (section of an arc) at specified horizontal and vertical offsets from the curve:

1. Define the curve. You can manually enter, or select from the list or map the following points:
   - In **PC Point**, enter the Point of Curve, the starting point of the arc.
   - In **PT Point**, enter the Point of Tangent to the curve, the ending point of the arc.
   - Select between **Radius/Chord Angle/Curve Angle** to enter the radius parameter of the curve as required. The **Length** of the curve will be shown on the dialog.
2. In **Start**, enter the curve station (chainage) to begin staking offsets.
3. From the **Height Computation** drop-down list select the type of height computations for the staked point:
   - *Ht of Start Pt* (the height of the start point): the staked point will have the same height as the PC Point of the curve.
   - *Interpolate Ht*: the height of the staked point will be computed through linear interpolation, using the height of the PC and PT Points of the curve.
4. In **Turn**, select the direction of turn, relative to the PC Point.
5. Select the **Include Transition Point** checkbox if the transition point should be included.
6. **Deviation Report** shows the name of the current stake report if it is set.
7. Click **Next** to proceed on the **Station & Offsets** dialog.

For the icons available on the dialog see **Icons Description**.

Layout 3-Pt Arc Offsets

To stake offsets from a curve defined by three points:

1. Define the curve. You can manually enter, or select from the list or map the following points:
   - In **PC Point**, enter the Point of Curve, the starting point of the arc.
   - In **PT Point**, enter the Point of Tangent to the curve.

- Select either the **Arc Pt** or the **RP Point** (the Radius Point). The dialog changes its appearance depending upon the selection. In the second case the distance between **RP Point** and **PC point** should be equal to the distance between **RP Point** and **PT point**. 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). Select either *Small* or *Large* from the **Arc** drop-down box to indicate which of these two curves should be used for computations.

2. Select the **Include Transition Point** checkbox if the transition point should be included.
3. In **Start**, enter the curve station (chainage) to begin staking offsets.
4. **Deviation Report**shows the name of the current layout report if it is set.
5. Click **Next** to proceed on the **Station & Offsets** dialog.

For the icons available on the dialog see **Icons Description**.

Layout Polyline

To stake a linework:

1. Select **Polyline / 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.
   - *Polyline*: 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. **Deviation 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 **Layout** to perform the stakeout. **More...**

Layout dialog

The **Layout** 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.

Layout 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 routines 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

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.

Plan View

The Plan 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 plan 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 Plan view and shows the current location and design location.

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.

Apps folder

Click an icon to use an application built in your device:

Take Photo
Starts the Camera application to take pictures. Not available for onboard.

Web Browser
Starts Internet Explorer to visit a web address you want. Not available for onboard.

Photo Viewer
Starts the Pictures & Videos application to view the pictures attached to the current job. Not available for onboard.

Files
Opens File Explorer to manage the files.

News
Allows you to view news and other information from Magnet RSS. Not available for onboard.

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.

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.