

MAGNET Office



Reference Manual



MAGNET Office Reference Manual

Part Number 1001299-01

Rev A

©Copyright Topcon Positioning Systems, Inc.

July, 2012

All contents in this manual are copyrighted by Topcon. All rights reserved. The information contained herein may not be used, accessed, copied, stored, displayed, sold, modified, published, distributed, or otherwise reproduced without the express written consent from Topcon.

Table of Contents

Preface	xv
Terms and Conditions	xv
Manual Conventions	xviii
Chapter 1	
Introduction	1-1
About This Manual	1-2
Chapter 2	
Getting Started	2-1
System Requirements	2-1
Running MAGNET Office	2-2
Setting up MAGNET Office with a Network USB Key .	2-4
USB Key Directory	2-4
ccNetKey Directory	2-4
Executable File for the MAGNET Office	
USB Key	2-5
Updating MAGNET Office on the USB Key	2-5
Installation Instructions for the Server	2-6
Installation Instructions for the Workstation	2-7
Startup Component Services	2-8
Chapter 3	
Using MAGNET Office	3-1
Using the Mouse	3-1
Defining Mouse Pointers	3-2
Using the Mouse Right-Click	3-3
Preventing Moving Entities	3-3
Using the Toolbar	3-6
Using the Status Bar	3-12
Selecting Data	3-13
Shortcut Menus	3-13

Using the Shortcut Keys 3-14
 Repeat a Command 3-14
 Reset the Current Command 3-15
Activating Zoom and Pan 3-15

Chapter 4

Working with Data Types 4-1
 Basic Data Types or Entities 4-1
 Defining a Point 4-1
 Defining a Line 4-4
 Defining Arcs and Circles 4-6
 Defining a String 4-7
 Defining a Polygon 4-8
 Creating Text 4-10
 Defining an Arrow 4-12
 Defining Dimensions 4-13
 Derived Data Types or Entities 4-14
 Creating Boundaries 4-14
 Defining Lots (Allotments) 4-15
 Creating Pads 4-17
 Defining Structures 4-17
 Triangles in Digital Terrain Models (DTM) 4-18
 Interpolating Contours 4-19
 Defining Breaklines 4-19
 Defining Curb Strings 4-20
 Designing Intersections 4-21
 Designing Alignments 4-21
 Viewing DTM in 3D 4-23
 Additional Data Types 4-23
 Defining Symbols 4-23
 Defining Line Styles 4-24
 Defining Title Blocks 4-25
 Defining Layers 4-26
 Configuring Angular Units Entry 4-26
 Defining Bearings and Angles 4-27
 Defining Distance Units Entry 4-28

Chapter 5

Survey View: File Menu 5-1

- About the Survey View 5-2
- File Menu Options 5-4
- New 5-5
- Open 5-7
 - Close 5-8
- Save 5-9
 - Save a Project File to Another Name 5-10
 - Project File Manager 5-10
- Import 5-12
- Export 5-19
- Export to Pocket-3D 5-25
- Genio Translation Table 5-26
 - Autocad Layer Translation Table 5-29
 - Synchronization Utility 5-31
 - Synchronization Settings 5-32
 - Desktop-CE Device Synchronization 5-35
 - Synchronization Setting 5-35
- Data Transfer 5-36
 - Print Setup 5-37
- Print 5-38
- Exit 5-39

Chapter 6

Survey View: Edit Menu 6-1

- Undo 6-2
- Redo 6-2
- Undo History 6-3
- Redo History 6-3
- Cut 6-4
- Copy 6-4
- Paste 6-6
- Delete 6-8
- Select Point 6-9
- Select By 6-10
 - Layers Tab 6-10
 - Points Tab 6-11

Table of Contents

Lines, Arcs, Circles, and Strings	6-13
Polygons Tab	6-15
Lots Tab	6-16
Text Tab	6-17
De-Select By	6-18
Select By Region	6-19
Select By Name	6-20
Select By DTM	6-21
Select All	6-22
Invert Selection	6-22
DeSelect All	6-23
Edit Properties	6-23
Repeat	6-24

Chapter 7

Survey View: View Menu	7-1
Redraw	7-2
Regen	7-2
Zoom	7-2
Pan	7-5
Save View	7-5
Toolbar Settings	7-7
Layers	7-7
Add Layers	7-10
Edit Layers	7-11
Control Layers	7-13
3D Viewer	7-15
Settings Menu	7-17
Unit	7-17
DTM Filling	7-18
Fill	7-18
Smooth	7-19
Speed	7-19
Vertical Exaggeration	7-19
Change Views	7-22
Status Bar Options	7-22
Ruler	7-23
Grid	7-23

Correct Positioning Data Using the Grid Command . 7-24

Chapter 8

Survey View: Settings Menu 8-1

- Project Settings 8-2
 - Project Details 8-3
 - Project Changes 8-4
 - Statistics 8-6
 - Unit Settings 8-7
 - Symbols 8-8
 - Coordinate Systems 8-9
- Program Settings 8-10
 - User Information 8-11
 - Folders 8-12
 - Libraries 8-13
 - Unit Settings 8-14
 - Symbols 8-17
 - Coordinate System 8-18
 - Preferences 8-19
- Create Custom Colors 8-22
- Display General Settings 8-23
- Default Attributes 8-26
- Text Styles 8-28
 - Add Text Styles 8-29
 - Modify Text Styles 8-30
 - Copy a Text Style From the Library 8-30
- Justification Styles 8-31
 - Add 8-32
 - Modify 8-34
 - Rename 8-34
 - Delete 8-34
 - Copy From Library 8-35
- Define Annotation Styles 8-35
 - Libraries 8-36
 - Display Annotations 8-37
 - Distance Annotation Settings 8-37
 - Angle Annotation Settings 8-38
 - Area Annotation Settings 8-40

Table of Contents

Dimension/Arrow Settings	8-41
Point Annotation Settings	8-43
Line Annotation Settings	8-45
Arc Annotation Settings	8-47
Lot Annotation Settings	8-49
Bearing Annotation Settings	8-51
Point Creation Mode	8-53
Point Coordinates Mode	8-54
Point Latitude and Longitude Mode	8-55
Point Radiation Mode	8-55
Create Points Mode	8-57
Apply the Snap Mode	8-57
Point Snap Mode	8-58
End Point Snap	8-59
Intersect Snap	8-59
Mid Point Snap	8-59
Arc Center Snap	8-59
Line Snap	8-59
Perpendicular Snap	8-60
Grid Point Snap	8-60
Shortcut Keys	8-60

Chapter 9

Survey View: Insert Menu	9-1
Create a Point	9-2
Radiation	9-3
XYZ	9-4
Offset/Distance	9-4
Offset/Angle	9-5
Intersection	9-6
Bearing/Distance Intersection	9-8
Lines	9-8
Define an Arc	9-19
Define a Circle	9-31
Polylines	9-33
Polygons	9-33
Rectangle	9-36
Strings	9-37

String	9-38
Parallel String	9-40
Parallel Figure	9-40
Define Text	9-43
Insert Text	9-44
Multiline Text	9-46
Bearing and Distance	9-47
Point Coordinate	9-48
Point Height	9-48
Arc Length	9-50
Arc Radius	9-51
Stacked Arc Labels	9-51
Angle	9-52
Custom Tables	9-53
Image	9-60
Dimension	9-64
Insert Arrow	9-65
Straight Arrow	9-65
Z Arrow	9-66
Keywords	9-67

Chapter 10

Survey View: Modify Menu 10-1

Edit Points	10-2
Multiple Point Edit	10-2
Set Point Height	10-5
Modify a Line	10-6
Segment Line	10-7
Extend or Trim Line	10-9
Multiple Trim	10-10
Extend All	10-12
Trim All	10-12
Move Point/Line	10-12
Insert Point Into Line	10-13
Delete Point on Line	10-14
Flip Line	10-15
Arc	10-15
Modify Arc	10-15

Table of Contents

Segment Arc	10-16
Convert Three Point Arc	10-17
Insert Point Into Arc	10-18
Delete Point on Arc	10-18
Flip Line	10-18
String	10-19
Add Point to String/Polygon	10-20
Delete Point in String/Polygon	10-20
Swap Points in String	10-21
Reverse String	10-21
Join Strings	10-22
Split String	10-23
Delete Segment	10-23
Intersect Strings	10-24
Convert String to Polyline	10-24
Convert Polyline to String	10-25
Create String From Lines	10-25
Set String Point Elevation	10-26
Text	10-27
Rotate	10-27
Align Text	10-28
Link	10-28
Unlink Text	10-29
Convert Annotation	10-29
Extend/Trim Selection to Boundary	10-30
Properties	10-31
Match Properties	10-31
Create Points from Entities	10-32
Delete Points from Entities	10-32
Explode	10-32
Filter Points	10-33
Copy	10-35
Move	10-36
Rotate	10-37
Align	10-37
Scale	10-40
Offset	10-40

Chapter 11

Survey View: Inquiry Menu 11-1

- Area 11-1
- Angle 11-3
- Bearing/Distance 11-4
- List Data 11-5
- List Project Details 11-6
- Inverse Report 11-7

Chapter 12

Survey View: Survey Menu 12-1

- Raw Data Editor 12-2
- Geodetic Calculator 12-2
- Display the Calculator 12-5
- Manual Data Entry 12-5
 - Point Entry 12-5
 - Traverse Editor 12-8
 - Deed Entry 12-13
 - Legal Description 12-17
- Control Points 12-21
- Translate/Rotate Data 12-22
 - Translate/Rotate Data 12-23
 - Select Data 12-25
- Traverse Adjustment 12-26
- LS (Least Squares) Network Adjustment 12-28
 - Managing/Processing LS Network Adjustments 12-30
- Using the Data Editor Observations and Points Tabs 12-33
 - Observations Tab 12-33
- Least Squares Transformation 12-41
- Define Structure 12-44
- Locate Structure 12-45

Chapter 13

Survey View: Design Menu 13-1

- Manual Road Entry 13-1
 - Create Surface 13-9
 - Create Road 13-10
- Level Book 13-12

Table of Contents

Defining Alignments	13-14
Manual Alignment Entry	13-16
Create Alignment	13-20
Create Alignment From Lines	13-23
Edit IP (Intersection Points)	13-24
Add IP	13-27
Insert IP	13-28
Delete IP	13-29
Alignment Text	13-29
Delete Alignment Text	13-31
Alignment Report	13-32
Offset Report	13-34
Convert Alignment to String	13-35
Create Center Line Points	13-36
Catch Points	13-37
Create Offset Strings	13-39
Create Multi Sheet Plot Windows	13-41
Intersection	13-43
Create Fillet Curb Return	13-44
Create Cul-de-sac	13-45
Trim String	13-45
Create Curb From String	13-47
Create an Island Curb	13-49
Create Intersection	13-52
Design Profiles	13-54
Profile From DTM	13-54
Profile From String	13-56
Profile From Points	13-57
Quick Profiles	13-59
Profile Editor	13-59
Profile Report	13-62
Quick Profile	13-62
Cross Sections Menu	13-63
Cross Section From DTM	13-63
Cross Sections From Strings	13-66
Cross Sections From Points	13-67
Quick Cross Section	13-69
Cross Section Editor	13-73

Cross Section Volume	13-75
Import Cross Section Files	13-75
Cross Section Report	13-76
Subdivision	13-76
Lots Layout	13-77
Lot By Interior Point	13-80
Show Lots	13-83
Lot Report	13-84
Annotation Lots	13-85
Annotate Lot Back Boundary	13-86
Annotate Lot Sides by Brg/Dist	13-87
Annotate Lot Sides By Distance	13-88
Delete Lot Annotation	13-90
Create Pad	13-90
Pad Design	13-93
Road View/Edit	13-99

Chapter 14

Survey View: Surface Menu	14-1
DTM Settings	14-2
Contour Settings	14-4
Create DTM (Digital Terrain Model)	14-5
Create DTM by Depth Range	14-14
Compute Elevations	14-15
Grid Points	14-16
Interpolate Point Elevations	14-17
Create Point by Grade	14-19
Set Elevation by Grade of Line	14-20
Elevation Difference	14-21
Set Point Elevation by Reference Elevation	14-22
Intersect Grades	14-24
Drape Object	14-25
Set Elevation by Intersection with Other Entities	14-25
Set Elevation From Text	14-26
Set Elevation by Increment	14-27
Set Start and End Elevation	14-28
Merge DTM	14-29
Update DTM	14-30

Table of Contents

Trim DTM	14-31
DTM Tools	14-31
Export DTM	14-33
Import DTM	14-33
Create Height Difference DTM	14-34
Raise/Lower DTM	14-35
Check DTM	14-36
3D View	14-37
3DViewer Screen	14-40
About 3DViewer	14-45
Add Triangle	14-45
Delete Triangle	14-46
Swap Triangles	14-52
Show Triangle Properties	14-52
Insert Breakline	14-53
Check Breaklines	14-56
Check Contourable Points	14-58
Decompose Triangles	14-59
Convert Contour to String	14-61
Create DTM Boundary	14-63
Create Boundary	14-63
Create Enclosing Boundary	14-67
Create Boundary From Polygons	14-70
Create Boundary List	14-71
Edit Boundary Lists	14-73
Compute Volume	14-74
Batch Volumes	14-80
Quick Volume	14-82

Chapter 15

Survey View: Tools Menu 15-1

Using the Drawing Editor	15-2
Symbol Editor	15-2
Line Style Editor	15-4
Create Plot Windows	15-6
Reposition Plot Window	15-8
Show Plot Windows	15-9
List Plot Windows	15-9

Bring Polygon to Front 15-9
 Modify an Annotation 15-10
 Annotation Table Settings 15-13
 Renumber Points 15-16
 Renumber String Points 15-19

Chapter 16

Survey View: Library Menu 16-1
 Set Up Survey Codes 16-2
 The Six Parts of the Code 16-3
 Survey Codes 16-8
 Code Settings 16-12
 Create Custom Colors 16-17
 Text Styles 16-18
 Modify 16-19
 Rename 16-20
 Delete 16-20
 Copy From Library 16-20
 Justification Styles 16-21
 New 16-22
 Modify 16-23
 Rename 16-24
 Delete 16-24
 Copy From Lib 16-24
 Annotation Styles 16-24
 Legal Description Templates 16-24
 Legal Description Template Library 16-27
 Define Template Components 16-28
 Symbols 16-29
 Add Symbols to a Library 16-30
 Edit a Symbol 16-33
 Rename a Symbol 16-33
 Delete a Symbol 16-34
 Import a Symbol 16-34
 Line Styles 16-34
 Create a Line Style 16-35
 Edit a Line Style 16-37
 Rename a Line Style 16-38

Table of Contents

Delete a Line Style	16-38
Import a Line Style	16-38
Plot Settings	16-39
Contour Settings	16-46
Contour Settings Parameters	16-47
Genio Translation Settings	16-48
Autocad Layer Definition Tables	16-50
Autocad Layer Translation Tables	16-52
Page Setup	16-55
Title Block	16-56
Create Plot Window	16-57
Setting Margins for the Title Block	16-58
Symbol	16-59
Define Line Style	16-61

Chapter 17

Survey View: Window Menu/Help Menu 17-1

New Window	17-1
Cascade	17-2
Tile Horizontally	17-3
Tile Vertically	17-3
Help Menu	17-4
Help	17-5
Display Hints	17-5
About Topcon MAGNET Office	17-5
Registration	17-6

Chapter 18

Roads View Editor 18-1

About the Road View Editor	18-1
Creating Alignments	18-5
Display Settings (Hot Button)	18-5
Selecting a Current String	18-6
Vertical Intersection Point	18-6
Displaying Leader Lines	18-6
Viewing the Current Surface	18-7
File Menu	18-7
Open	18-8

Save	18-8
Cross Section Editor	18-9
Edit Menu	18-9
Cut	18-10
Copy	18-10
Paste	18-12
View Menu	18-12
Redraw	18-13
Regen	18-13
Zoom	18-14
Pan	18-14
Toolbar	18-15
Status Bar	18-20
Return to Survey View	18-20
Settings Menu	18-21
Alignment Pane	18-21
Profile Pane	18-23
CrossSection Pane	18-25
Alignment Menu	18-27
Add String	18-28
Delete String	18-31
Update Survey	18-31
Data Transfer Table	18-31
Curve Report	18-34
Profile Menu	18-34
Add IP	18-35
Add IP by Grade	18-37
Add IP by Intersection	18-38
Add IP by TP	18-39
Raise Design	18-39
Compute VC's	18-40
Compute Resheet Levels	18-43
Resheet Parameters	18-45
Compute Level Differences	18-47
Profile Report	18-48
VC Report	18-48
CrossSection Menu	18-50
Add Surface	18-51

Table of Contents

Additional Information	18-53
Subgrade Sequence	18-53
Delete Surface	18-54
Add Point	18-55
Raise Point	18-56
Adjust Side Slope	18-57
Intersect Side Slope	18-58
Strip Depth	18-59
Volume Factors	18-60
Omit Stations	18-61
Compute Volumes	18-62
Additional Information for Computing	
Volumes	18-64
Updating Changes in Survey View	18-64
Cross Sections Report	18-65
Construction Report	18-65
Design Menu	18-67
Templates	18-68
Super Elevation Tables	18-70
Create Super Table	18-73
Compound Side Slopes	18-78
Symbol Table	18-81
Template Selection	18-82
Super Table	18-83
Design Section Selection	18-85
Subgrade Template Selection	18-88
Subgrade Super Selection	18-91
Subgrade Section Selection	18-92
Pavement Parameters	18-93
Side Slopes	18-96
Fixed Offset Slopes	18-98
Compound Side Slopes	18-100
Ditches	18-102
Shoulder Extension	18-104
Compute Design	18-105
Plotting Menu	18-107
Profile Plot Settings	18-107
Cross Section Plot Settings	18-113

Plot Standard Profile	18-120
Plot Offset Profile	18-121
Plot CrossSection	18-122
Library Menu	18-123
Profile Plot Table	18-124
Cross Section Plot Table	18-127
Sight Distance Table	18-131
Friction Factors	18-133
Curve Radius	18-134
Window Menu	18-135
Help Menu	18-135

Preface

Thank you for purchasing this Topcon product. The materials available in this Manual (the “Manual”) have been prepared by Topcon Positioning Systems, Inc. (“TPS”) for owners of Topcon products, and are designed to assist owners with the use of the receiver and its use is subject to these terms and conditions (the “Terms and Conditions”).

Note: Please read these Terms and Conditions carefully.

Terms and Conditions

USE This product is designed to be used by a professional. The user should have a good knowledge of the safe use of the product and implement the types of safety procedures recommended by the local government protection agency for both private use and commercial job sites.

COPYRIGHT All information contained in this Manual is the intellectual property of, and copyrighted material of TPS. All rights are reserved. You may not use, access, copy, store, display, create derivative works of, sell, modify, publish, distribute, or allow any third party access to, any graphics, content, information or data in this Manual without TPS’ express written consent and may only use such information for the care and operation of your receiver. The information and data in this Manual are a valuable asset of TPS and are developed by the expenditure of considerable work, time and money, and are the result of original selection, coordination and arrangement by TPS.

TRADEMARKS MAGNET Office, Topcon® and Topcon Positioning Systems™ are trademarks or registered trademarks of TPS. Windows® is a registered trademark of Microsoft Corporation. The

Bluetooth® word mark and logos are owned by Bluetooth SIG, Inc. and any use of such marks by Topcon Positioning Systems, Inc. is used under license. Other product and company names mentioned herein may be trademarks of their respective owners.

DISCLAIMER OF WARRANTY EXCEPT FOR ANY WARRANTIES IN AN APPENDIX OR A WARRANTY CARD ACCOMPANYING THE PRODUCT, THIS MANUAL AND THE RECEIVER ARE PROVIDED “AS-IS.” THERE ARE NO OTHER WARRANTIES. TPS DISCLAIMS ANY IMPLIED WARRANTY OF MERCHANTABILITY OR FITNESS FOR ANY PARTICULAR USE OR PURPOSE. TPS AND ITS DISTRIBUTORS SHALL NOT BE LIABLE FOR TECHNICAL OR EDITORIAL ERRORS OR OMISSIONS CONTAINED HEREIN; NOR FOR INCIDENTAL OR CONSEQUENTIAL DAMAGES RESULTING FROM THE FURNISHING, PERFORMANCE OR USE OF THIS MATERIAL OR THE RECEIVER. SUCH DISCLAIMED DAMAGES INCLUDE BUT ARE NOT LIMITED TO LOSS OF TIME, LOSS OR DESTRUCTION OF DATA, LOSS OF PROFIT, SAVINGS OR REVENUE, OR LOSS OF THE PRODUCT’S USE. IN ADDITION TPS IS NOT RESPONSIBLE OR LIABLE FOR DAMAGES OR COSTS INCURRED IN CONNECTION WITH OBTAINING SUBSTITUTE PRODUCTS OR SOFTWARE, CLAIMS BY OTHERS, INCONVENIENCE, OR ANY OTHER COSTS. IN ANY EVENT, TPS SHALL HAVE NO LIABILITY FOR DAMAGES OR OTHERWISE TO YOU OR ANY OTHER PERSON OR ENTITY IN EXCESS OF THE PURCHASE PRICE FOR THE RECEIVER.

LICENSE AGREEMENT Use of any computer programs or software supplied by TPS or downloaded from a TPS website (the “Software”) in connection with the receiver constitutes acceptance of these Terms and Conditions in this Manual and an agreement to abide by these Terms and Conditions. The user is granted a personal, non-exclusive, non-transferable license to use such Software under the terms stated herein and in any case only with a single receiver or single computer. You may not assign or transfer the Software or this license without the express written consent of TPS. This license is effective until terminated. You may terminate the license at any time by destroying

the Software and Manual. TPS may terminate the license if you fail to comply with any of the Terms or Conditions. You agree to destroy the Software and manual upon termination of your use of the receiver. All ownership, copyright and other intellectual property rights in and to the Software belong to TPS. If these license terms are not acceptable, return any unused software and manual.

CONFIDENTIALITY This Manual, its contents and the Software (collectively, the “Confidential Information”) are the confidential and proprietary information of TPS. You agree to treat TPS’ Confidential Information with a degree of care no less stringent than the degree of care you would use in safeguarding your own most valuable trade secrets. Nothing in this paragraph shall restrict you from disclosing Confidential Information to your employees as may be necessary or appropriate to operate or care for the receiver. Such employees must also keep the Confidentiality Information confidential. In the event you become legally compelled to disclose any of the Confidential Information, you shall give TPS immediate notice so that it may seek a protective order or other appropriate remedy.

WEBSITE; OTHER STATEMENTS No statement contained at the TPS website (or any other website) 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, warranty and limitation of liability).

SAFETY Improper use of the receiver can lead to injury to persons or property and/or malfunction of the product. The receiver should only be repaired by authorized TPS warranty service centers. Users should review and heed the safety warnings in an Appendix.

Mercury Warning The LCD display in the 9168 or GX-60 Topcon Control Box contains mercury. The Control Box should not be disposed of or placed in a waste stream destined for disposal until the mercury is either removed and reused, recycled, or otherwise managed to ensure that the mercury in the product does not become mixed with other solid waste or wastewater.

MISCELLANEOUS The above Terms and Conditions may be amended, modified, superseded, or canceled, at any time by TPS. The

above Terms and Conditions will be governed by, and construed in accordance with, the laws of the State of California, without reference to conflict of laws.

Manual Conventions

This manual uses the following conventions:

Example	Description
File ▶ Exit	Click the File menu and click Exit .
<i>Connection</i>	Indicates the name of a dialog box or screen.
<i>Frequency</i>	Indicates a field on a dialog box or screen, or a tab within a dialog box or screen.
Enter	Press or click the button or key labeled Enter .



Further information to note about the configuration, maintenance, or setup of a system.



Supplementary information that can help you configure, maintain, or set up a system.



Supplementary information that can have an affect on system operation, system performance, measurements, or personal safety.



Notification that an action has the potential to adversely affect system operation, system performance, data integrity, or personal health.



Notification that an action *will* result in system damage, loss of data, loss of warranty, or personal injury.



Under no circumstances should this action be performed.

Notes:

Introduction

MAGNET Office is CAD software developed by Topcon for use by surveying professionals. With its extensive range of tools and options, MAGNET Office provides a complete desktop survey and drafting solution.

MAGNET Office works seamlessly with Topcon products and is format compatible with most other programs, like AutoCAD, providing numerous import and export options.

With MAGNET Office, you can display survey data, line work, and points as the data is entered, work with multiple drawings on different scales within the same project, and perform calculations, adjustments, and point creation in the survey view and drafting functions in the drawing view.

MAGNET Office is used for:

- Data transfer and processing
- Graphical data displays
- Reports and legal descriptions
- Survey computations
- Lot design
- Surface generation
- Road alignments
- Cross sections and profiles
- Deed entry
- Legal descriptions
- Manual survey data entry
- Traverse Adjustment

- MAGNET Office Graphical cogo routines
- Automated pad design with automatic cut/fill balance

Project Files The MAGNET Office project file is self-contained and space-efficient. It provides a facility to store any files related to a project into a single file, making tasks like archiving simple.

A project file can contain all survey data, raw data, surfaces, roads and drainage designs and drawings. Required library data is also saved in the project file to ensure portability.

Flexibility MAGNET Office offers numerous options for subdivision, road, and drainage design, and its comprehensive editing capabilities enable you to reprocess survey data and update graphics immediately. In addition, MAGNET Office is extremely customizable, so you can control how a project is finished.

About This Manual

This is the reference manual for MAGNET Office software. It is intended to describe how to install and operate MAGNET Office. Unless otherwise stated, the operational characteristics described herein correspond to factory default configurations and settings as shipped from Topcon Positioning Systems.

Getting Started

MAGNET Office comes pre-installed on a customized USB key with password capability. You can install MAGNET Office on your computer or run MAGNET Office directly from the USB drive.

Unlicensed users can still download and operate MAGNET Office as a demo, during which time, they will not be able to save or print any data.

This chapter describes how to get MAGNET Office running on your computer:

- “System Requirements”
- “Running MAGNET Office” on page 2-2
- “Setting up MAGNET Office with a Network USB Key” on page 2-4
- “Updating MAGNET Office on the USB Key” on page 2-5

System Requirements

To determine if your computer can run the MAGNET Office software, make sure it is compatible with the specifications listed in Table 2-1.

Table 2-1. Hardware Specifications

Components	Minimum Requirement
Computer CPU	Pentium processor, 1 GHz or higher
RAM	512 MB
Operating System	Windows 2000, XP, Vista
USB Port	USB 2.0
Monitor	1024 x 768 res.
Mouse	Scroll wheel

Running MAGNET Office

To run MAGNET Office, do the following:

1. Insert the customized USB key into a USB-2 port.
A selection dialog box appears (Figure 2-1).



Figure 2-1. MAGNET Office Setup Window

2. Make sure MAGNET Office is highlighted and click **OK**.
3. On the **Application Startup** window, click **Start**.
The **Startup Wizard** appears (“Startup Wizard” on page 2-3).

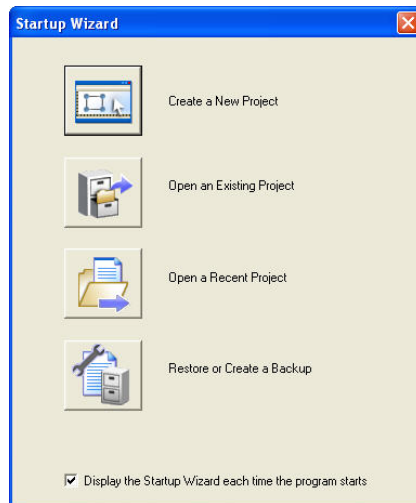


Figure 2-2. Startup Wizard

4. Make a selection, and follow the prompts to display the project area.



Installation of the MAGNET Office software confirms the user's acceptance of the terms and conditions of the license agreement to use MAGNET Office as granted by Topcon Positioning Systems, Inc.


Click the *MAGNET Office* icon  on the desktop to start the software. When MAGNET Office first starts up, you are advised to follow the process for checking the software setup. This ensures that work done within MAGNET Office is accessing the correct libraries and using various software options effectively. The paths displayed in Table 2-2 are the defaults used by the installation but may be different on your computer.

Table 2-2. Folder Paths

Folder Name	Full Path
MAGNET Office Program	c:\xxx
AddOns	c:\xxx\AddOns

Table 2-2. Folder Paths (Continued)

Folder Name	Full Path
CoordSys	c:\xxx\CoordSys
Data	c:\xxx\Data
Help	c:\xxx\Help
Library	c:\xxx\Library
Notes	c:\xxx\Notes
Programs	c:\xxx\Programs
Temp	c:\xxx\Temp
Tools	c:\xxx\Tools



Do not delete the License folder.



The license for MAGNET Office is embedded on the customized USB key.



Please visit www.topcompositioning.com for frequent software updates to MAGNET Office.

Setting up MAGNET Office with a Network USB Key

The MAGNET Office network USB key contains two directories and one file.

USB Key Directory

License – Licensing control directory

ccNetKey Directory

- Licensing control program directory

- RunAsService (Utility program to make ccNetKey run as a service)
- The Control Program, ccNetKey must be executed for licensing control

Executable File for the MAGNET Office USB Key

Executable File – ccSetupVVV.exe

Updating MAGNET Office on the USB Key

Throughout the year, you can download MAGNET Office updates from www.civilcad.com.au/d-load/smSetup.exe. The smSetup.exe files is for several purposes, including: USB key updates, existing installation updates, and new installation. Administrator status is not required (Figure 2-3).

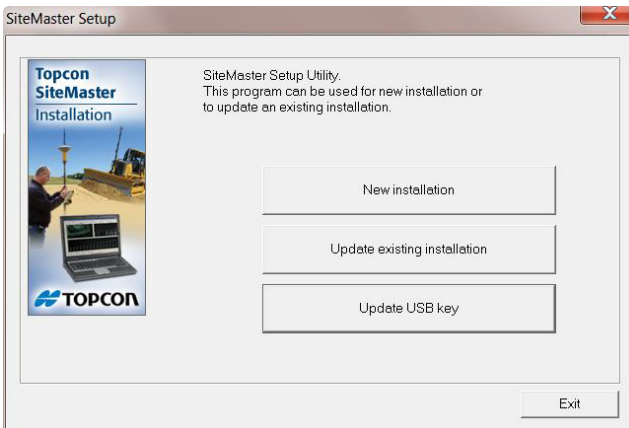


Figure 2-3. MAGNET Office Setup Screen – Update MAGNET Office

Installation Instructions for the Server

Insert the MAGNET Office network USB key into a USB2 port of the designated computer. The computer must have the ability to share this USB key as a resource. Once the USB key is recognized by the computer, share it as “NetKey” with READ ONLY access (Figure 2-4).

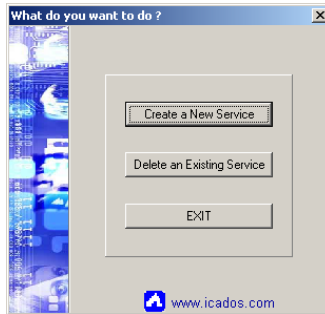


Figure 2-4. Create A New Service

ccNetKey must be installed as a service in order to startup automatically. The steps described below are just one of many methods available for setting an application as a service.

1. From the ccNetKey directory, execute RunAsService.
2. Click **Create a New Service**, then select *ccNetKey.exe* file.
3. Complete the instructions shown in the *Service configuration* dialog box in Figure 2-5 on page 2-7 and click **Create Service**.
4. Click the **Control Panel ▶ Administrative Tools**.

Library Files You can relocate the library file to a server for sharing with other users. The libraries installed during setup are sample files. You may have other sample library files available to them provided by the local MAGNET Office agent to match country or state requirements, or existing library files already used within the company, organization, or council.

Data Files Users can relocate the data file to a server for sharing data files with other users and to ensure regular backups. Also,

subdirectories should be available to sort data by different project names, client names, user names, or other category.

Temporary Files This folder is usually retained on the workstation.

Installation Instructions for the Workstation

Do the following to install MAGNET Office at your Workstation:

1. Navigate to the MAGNET Office Network USB key by typing in \\yourservername\NetKey and double-click on **ccSetup** to install MAGNET Office onto your local workstation.

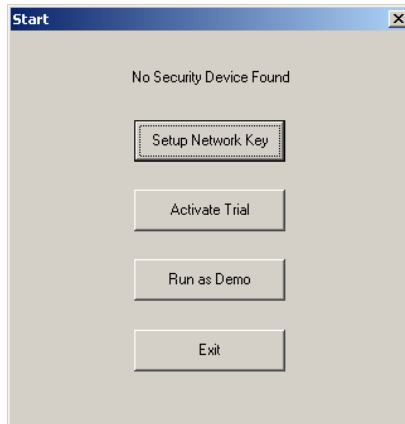


Figure 2-5. The Start Screen

2. Start MAGNET Office and click **Setup Network Key** (Figure 2-5).
3. Enter your server name, \\yourservername or start MAGNET Office in Demo Mode.
4. Using the option, **File ▶ Program Settings**, enter the network USB key resource name \\yourservername\NetKey in the Network Key Directory field, located at the bottom of the *Directories* tab.
5. Click **OK** and restart MAGNET Office.

6. When prompted to register the workstation, click **Register Site** and follow the prompts.

On successful registration, you will be prompted to select a MAGNET Office license to operate.

Repeat the above steps for all workstations as required or use the license control program, ccNetKey, to register more workstations.

The Site ID is generated from the MAC address of workstation's network card.

Startup Component Services

1. Locate the service, ccNetKey, and double-click to display ccNetKey service properties.
2. Click on the **Log On** tab and activate. Allow the service to interact with desktop (Figure 2-6 on page 2-9).
3. To start the service, select the **General** tab and click **Start**.



For the licensing control program to begin, ccNetKey should start up and take a minute or two to scan the network environment.

4. When the *ccNetKey Properties* screen displays. Click **Change** to create your own Password (Figure 2-6 on page 2-9).

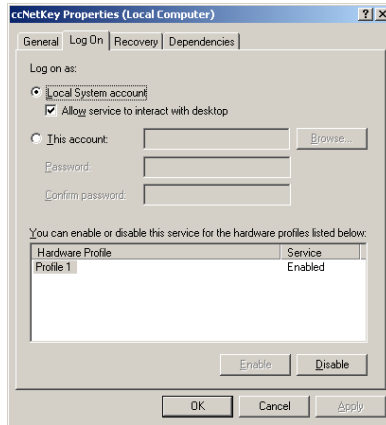
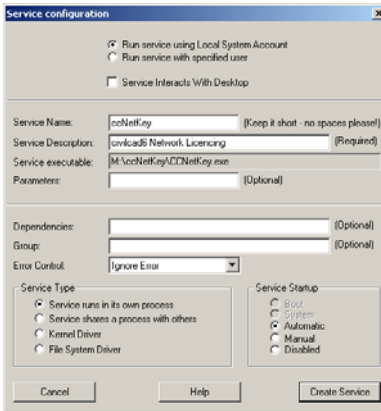


Figure 2-6. Service Configuration Screens

Using MAGNET Office

MAGNET Office is designed with several shortcut features that help you complete your tasks quickly and efficiently. This chapter shows you those standard features, tells you what they mean, and how to use them:

- “Using the Mouse”
- “Using the Toolbar” on page 3-6
- “Using the Status Bar” on page 3-12
- “Selecting Data” on page 3-13
- “Shortcut Menus” on page 3-13
- “Using the Shortcut Keys” on page 3-14
- “Activating Zoom and Pan” on page 3-15

Using the Mouse











The pointer appears as an arrow, a cross, or other shape, depending on the option selected and its position on the screen. As the pointer is moved about the screen, its position displays as Easting and Northing coordinates on the status bar on the lower left of the display area. These coordinates may be used for a new point when the left mouse button is used to select the insertion or computation option.

Use the mouse to select either an object, such as an entity, or a menu command. You can also use the left mouse button as the **Enter** key on the keyboard to accept data input. Double-clicking an entity opens that entity’s *Edit Properties* dialog box.

Defining Mouse Pointers

MAGNET Office uses a variety of pointer shapes to signify various actions. See Table 3-1 for details.

Table 3-1. Pointer Shapes

Pointer	Action
	The Standard pointer for selecting menu commands, buttons, toolbars, icons, and data entry fields in any dialog box.
	The Selection pointer displays in the drawing area of a View. Selects individual objects or drags a selection box around multiple objects.
	This pointer displays when using the insert commands. This pointer is often connected to a previously selected object with a construction stringline.
	The Insert Text pointer displays during a text insertion routine or when entering data into a field.
	The Move Entity pointer displays when the mouse is used to grab and move an object.
	The Rotate Entity pointer displays when the mouse is used to grab and change the orientation of an entity.
	The Rotate Text pointer displays at the opposite end of a selected text string to the text insertion point.
	The Zoom Window pointer displays when specifying a window for the Zoom Window option.
	The Pan pointer displays when using the Pan command to change the position of the data on the screen.
	The Star pointer indicates there is data selected. The number of objects selected is displayed with 0: nn next to the rolling coordinates in the Status bar.

Using the Mouse Right-Click

The right mouse button has several functions in MAGNET Office. Use the right button to:

- Configure a toolbar – Move the pointer over a toolbar and right-click to access the toolbar configuration window.
- Open the shortcut menu – Move the pointer anywhere within the display area, and right-click to open the menu of commands. For more information, see “Shortcut Menus” on page 3-13.

Preventing Moving Entities

To prevent entities from being moved or rotated inadvertently, press and hold the SHIFT key before you click and drag the entity to a new position.

When SHIFT is pressed and the mouse is positioned over a highlighted handle of the entity, either the Move Entity pointer or the Rotate Entity pointer displays. The left and right Shift keys produce different results, depending on the entities. See Table 3-2 below for more information.

Table 3-2. Moving Entities

Item	Action	Procedure
Point	To move a point:	<ul style="list-style-type: none"> • Press and hold either Shift key. • Click and drag the point.
Line	To move a line:	<ul style="list-style-type: none"> • Press and hold the left Shift key. Click and drag the handle at the line mid point. • Press and hold the right Shift key to display the Move Object pointer. Click and drag one of the three line handles.
	To rotate a line:	<ul style="list-style-type: none"> • Press and hold the left Shift key. • Click and drag the handle at the line end point.

Table 3-2. Moving Entities (Continued)

Item	Action	Procedure
Arc	To change the radius:	<ul style="list-style-type: none"> • Press and hold the left Shift key. • Click and drag the handle at the mid point of the arc length.
	To change the length:	<ul style="list-style-type: none"> • Press and hold the left Shift key. • Click and drag the handle at either arc end point.
	To move the center point:	<ul style="list-style-type: none"> • Press and hold the left Shift key. • Click and drag the handle at the arc center point. The arc start and end positions are held.
	To move the arc:	<ul style="list-style-type: none"> • Press and hold the right Shift key. • Click and drag one of the four arc handles.
Circle	To change the radius:	<ul style="list-style-type: none"> • Press and hold the left Shift key. • Click and drag the handle on the circle circumference.
	To move the circle:	<ul style="list-style-type: none"> • Press and hold the right Shift key. • Click and drag one of two circle handles.
String	To move a node:	<ul style="list-style-type: none"> • Press and hold the left Shift key. • Click and drag the string handle at the node point to move it and change its shape.
	To move the string:	<ul style="list-style-type: none"> • Press and hold the right Shift key. • Click and drag any of the string nodes to move the whole string without changing its shape.

Table 3-2. Moving Entities (Continued)

Item	Action	Procedure
Polygon, Boundary, Platform, or Lot	To move a node:	<ul style="list-style-type: none"> • Press and hold the left Shift key. • Click to select the polygon, and drag the handle at the node point to move the node and change the shape of the polygon.
	To move a polygon:	<ul style="list-style-type: none"> • Press and hold the left Shift key. Click and drag the polygon centroid point to move it. • Press and hold the right Shift key. Click and drag any point of the polygon to move it.
Alignment	To move an HIP:	<ul style="list-style-type: none"> • Press and hold the left Shift key. • Click and drag the HIP position to move it and change the alignment shape.
	To move the alignment:	<ul style="list-style-type: none"> • Press and hold the right Shift key. • Click and drag any HIP position to move the whole alignment without changing its shape.
Arrows	To move the end:	<ul style="list-style-type: none"> • Press and hold the left Shift key. • Click and drag one end of the arrow to rotate it.
	To move the arrow:	<ul style="list-style-type: none"> • Press and hold the right Shift key. • Click and drag either end of the arrow.
Dimension	To move a dimension:	<ul style="list-style-type: none"> • Press and hold the left Shift key. • Click and drag any of the three dimension handles. A dimension can only be moved toward or away from the object it has measured.
Text	To move text:	Click the text insertion point, and drag it.
	To rotate text:	Click and drag the text rotation handle.

Using the Toolbar

To display toolbar icons, do the following:

- Click **View ▶ Toolbar** to display the following icons: Properties, Text, Standard, Survey Tools, Add Entity, Modules, Align, Point Information, Snap Modes, and Input Modes.
- On the *Configure Toolbars* dialog box, check mark the box to the left of each toolbar icon you want to display on screen.
- Uncheck mark the toolbar icons you want to hide.

For more information the toolbar icons, see Figure 7-19 on page 7-20.

Table 3-3 describes the available toolbar icons.

Table 3-3. MAGNET Office Toolbar Icons


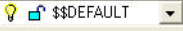





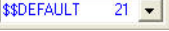
Icon	Icon Description
Properties Text Toolbar Icons	
	Displays the <i>Layer Settings</i> dialog box to change layer attributes, i.e., color, mark, symbol, line/width type, text style, etc.
	From the drop-down box, select a name for the current layer.
	Copies Settings to selected layer.
	Sets Properties from selected layer.
	Sets all attributes to selected bylayer.
	Restores previous settings for selected layer.
	Set the following attributes for the point from here: Point color, point mark, point symbol, point line style, text height, point code, etc.
Text Toolbar Icons	
	Click on the drop-down box to select a layer name.

Table 3-3. MAGNET Office Toolbar Icons (Continued)


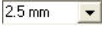















Icon	Icon Description
	Select a font style from the drop-down box.
	Select the line spacing from the drop-down box.
	Select either Bold, Initial, or Underline for the text style.
Standard Toolbar Icons	
	Creates a new file.
	Opens a file.
	Saves a file.
	Prints a file.
	Cuts text.
	Copies text.
	Pastes text.
	Erases (deletes) text.
	Undo previous action.
	Redo previous action.
	Repeats an action.
	Refreshes the display of data on screen.
	Rebuilds the screen display from the project database.
	Click this icon to display the ruler. Click again to hide the ruler.

Table 3-3. MAGNET Office Toolbar Icons (Continued)











Icon	Icon Description
	Use Zoom All to view the all data on screen.
	<p>Use Zoom Window to map a rectangular area of the data in one of the three panes to the extent of the pane.</p> <p>The prompt line, “<i>Locate First Point of Window</i>”, displays.</p> <ul style="list-style-type: none"> Click on one corner of the window. The prompt, “Locate 2nd Point of Window”, displays. Click at the opposite diagonal of the rectangular window. <p>The entities in the window will be mapped to fit in the display area. Simply draw a selection window in the current pane of the View. The display maps the data in the extents of the window to the size of the pane.</p>
	<p>Use Zoom Box to map an area of selected data within a box to the extent of the pane.</p> <ul style="list-style-type: none"> Click the right-mouse button and select Zoom Box. A yellow box displays on screen and the prompt line, “<i>Locate Zoom Point</i>”. The data in the center of the box is mapped to the extents of the display area. To change the size of the box use the +/- keys. To cancel this option, right-click and select Cancel or press the Esc key.
	Zoom in increases the magnification of the design view each time you press the button. The zooming pivot is the center of the screen.
	Zoom out decreases the magnification of the design view each tim you press the button. The zooming pivot is the center of the screen.
	Zooms in to the selected point .
	Zooms in to the previous point .
Survey Tools Toolbar Icons	
	Bearing/Distance Intersection
	Insert Point into Line
	Delete Connecting Point

Table 3-3. MAGNET Office Toolbar Icons (Continued)




















Icon	Icon Description
	Segment Line, Arc, or Circle
	Extend or Trim Line
	Offset Line, Arc, or Circle
	Add TP-IP
	Splay Corner
	Fillet Arc
	Enquire Point
	Enquire Bearing and Distance
	Enquire Offset Distance
	Query Area
	Translate/Rotate
Add Entity Toolbar Icons	
	Adds a Point
	Creates a Point Offset
	Creates a Point Intersection
	Adds a Line
	Join Points on a line.
	Creates a Perpendicular line.
	Creates a Parallel line.
	Creates a Tangent

Table 3-3. MAGNET Office Toolbar Icons (Continued)


















Icon	Icon Description
	Creates an External Tangent
	Creates an Arc – Angle
	Creates an Arc – Radius
	Creates an Arc – 3 Points
	Creates a Circle – Radius
	Creates a String
	Creates a Polygon
	Inserts a Rectangle
	Inserts Text
	Inserts Multiline Text
	Creates an Arrow
	Creates Dimensions
Modules Toolbar Icons	
	Displays the Survey view.
	Displays the Drawing view.
	Displays the Road view.
	Displays the Drainage view.
	Displays the Intersection view.
	Displays the Raw Data Editor view.
	Displays the Sewer view.

Table 3-3. MAGNET Office Toolbar Icons (Continued)























Icon	Icon Description
NOTE: The options active on the toolbar depends upon the hardware lock configuration.	
Align Toolbar Icons	
	Align Left
	Aligns text to the Right of the screen.
	Aligns text to the Top of the screen.
	Aligns text to the Bottom of the screen.
	Rotates text
Point Information Toolbar Icons	
	Displays point numbers.
	Displays point heights.
	Displays point codes.
	Displays contour points.
Snap Modes Toolbar Icons	
	Snaps to the nearest point.
	Snaps to the end point.
	Snaps to the intersection point.
	Snaps to the mid point.
	Snaps to the center of the arc.
	Snaps to the line.
	Snaps to the perpendicular line.
	Snaps to the grid point.







Table 3-3. MAGNET Office Toolbar Icons (Continued)

Icon	Icon Description
	Displays Drainage view.
Input Modes Toolbar Icons	
	Set Point Radiation
	Set Point Coordinate
	Set Point Latitude and Longitude
	Set Point Creation

Using the Status Bar

The Status Bar, located at the bottom of the screen, holds several options as shown in Table 3-4 below.

Table 3-4. Status Bar Icons

		C 303.942, L -7.009	M S N Z	NUM
The Status Bar has the following options:				
	Displays the DTM dialog box to change attributes for the digital terrain model. NOTE: Not active in Roads View.			
	Displays the General Settings dialog box that controls the display of roads data on screen.			
C 268.984, L -0.142				
Displays the cursor position of the mouse. The chainage and Level display in meters in the Profile pane, the Offset and Level position display in the Cross Section pane, or the Easting/Northing coordinates display in the Alignment pane.				
	The units that are set for the project.			
	Displays when the Num Lock key is activated on the keyboard.			

Selecting Data

To select data, do one of the following:

- Click to select an entity, which then becomes highlighted.
- Click and drag the pointer window from left to right to select all of the entities on the active layers that are fully enclosed in the window.
- Click and drag the pointer window from right to left to select all of the entities that are fully enclosed or crossed by the window.

Shortcut Menus

Shortcut menu options vary according to the current activity and provides options for changing the behavior of a command (Figure 3-1 on page 3-13).

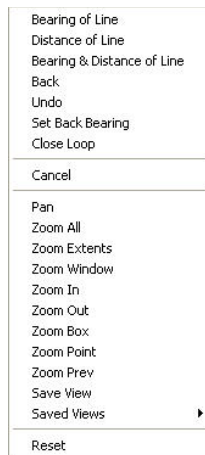


Figure 3-1. Shortcut Menu Example

Most shortcut menus contain the **Pan** and **Zoom** commands. When an entity is selected, the right-click menu changes. When there is more than one entity within the pointer snap area, the closest entity is selected. To select another object, right-click to access the **Next Selection** and **Previous Selection** commands.

Using the Shortcut Keys

The shortcut keys are listed in Table 3-5 below.

Table 3-5. Shortcut Keys

Keys	Action
Ctrl+N	Create a new project
Ctrl+O	Open a file
Ctrl+S	Save data
Ctrl+P	Print data
Ctrl+Z	Undo the last command
Ctrl+Y	Redo the last undone command
Ctrl+X	Cut selected data, and add it to the clipboard
Ctrl+C	Copy selected data to the clipboard
Ctrl+V	Paste data from the clipboard
Ctrl+A	Select all of the entities on the current layer
Ctrl+L	Open the Layer Settings Manager
Del	Delete selected items
Alt+Enter	Edit properties of a selected entity
F10	Display the plot windows

Repeat a Command

Once MAGNET Office performs a command, or even if it is canceled, the command remains available for use by selecting **Repeat** from the shortcut menu or the toolbar.

Reset the Current Command

The Reset command is available on the shortcut menu. Use this command to reset the current command, and start it again. You can use **Reset**, for example, to select new start points.

Activating Zoom and Pan

You can use the scroll wheel on the mouse to activate zoom and pan functions:

- **Zoom In** – Roll the scroll wheel away from you. The center of the Zoom In feature is based on the pointer position.
- **Zoom Out** – Roll the scroll wheel toward you. The center of the Zoom Out feature is based on the pointer position.
- **Zoom Extents** – Double-click the scroll feature.
- **Pan** – Press and hold the wheel and drag the pointer to another position.

Notes:

Working with Data Types

This chapter describes the following data types used in MAGNET Office and the attributes and properties associated with them:

- “Basic Data Types or Entities”
- “Derived Data Types or Entities” on page 4-14
- “Additional Data Types” on page 4-23

Each project is a whole, self-contained database, comprising a number of simple and complex entities.

Basic Data Types or Entities

All data consists of basic data types or entities, and each of these data types is stored in the database with varying attributes and properties.

Defining a Point

A point is a two- or three-dimensional position in space and is the most important entity from which all other entities are derived. Points can be computed, manipulated, edited, and deleted using a range of routines throughout MAGNET Office. Points are also computed when lines, arcs, strings, and other entities are created (Figure 4-1 on page 4-2).

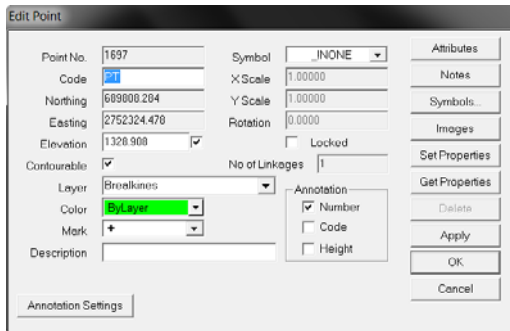



Figure 4-1. Edit Point

Points are defined in the database by the following parameters:

- Point Number – A unique identifier that distinguishes each point within a database. No two points can have the same point number or name in one database (project). Point numbers or names may be numeric or alphanumeric. The number or name is allocated when the point is created, either automatically by MAGNET Office or manually.
- Code – The code is an optional alphanumeric description attached to the point. Point codes are usually feature codes assigned to points during a survey. These feature codes are used to automatically assign properties to each point, as well as to control the line string between points during the reduction process. The feature codes and the properties are held in a user-defined library called the Survey Codes Library.
- Northing (N) – The north or Y component of a point’s position in space.
- Easting (E) – The east or X component of a point’s position in space.
- Elevation – The height, reduced level, or Z component of a point’s position above an arbitrary datum. MAGNET Office allows points to be created with or without a height. Check mark the elevation field box to activate it. An unchecked box indicates a point with no height.

- **Contourable** – A point that has a defined elevation may be available for inclusion in a digital terrain model (DTM). The contourable flag may be on (active) or off (inactive) for each point.
- **Layer** – Each point resides on one nominated layer.
- **Color** – Every point is assigned a color from the color palette. There are 16 standard colors and user-defined color options. Colors may be saved to the *Color.dat* file in the program folder
- **Mark** – These define the appearance of each point on the screen. There are several point mark styles available. The marks may be allocated a size for plotting or may be omitted from the plot if preferred.
- **Description** – Enter a description for the point (optional).
- **Symbol** – Symbols define the appearance of each point both on the screen and when plotted. They may be optionally assigned to a point. You can choose a symbol from a library of user-defined symbols, which are identified by an alphanumeric name. The symbols only display on the screen when the symbol check box is selected on the *Multiple Point Edit* dialog box or the *Hot Button* options. The **Hot Button**  is the small red rectangle on the status bar on the lower-left side of the display area.
- **X/Y Scale** – Each symbol is by default set to a scale of 1.00. Most symbols plot at the millimeter size they were designed at in the symbol library. These are unscalable symbols. The X scale and Y scale fields allow you to change the size of the symbol by multiplying the symbol's default size by the values in these fields. You can set different values for the X scale and Y scale for all symbols except circular symbols.

If the symbol is a scalable symbol, then it will plot at the scale specified by these X and Y scale values.
- **Rotation** – Each symbol is by default set to no rotation or 0 degrees. Rotations are absolute, and a value in DDD.MMSS set in this field will rotate the symbol by this angle.

- **Locked** – A point may be locked so its properties and attributes may not be edited. A locked point may not be moved with the mouse.
- **No. of Linkages** – The point is the basic data type from which all other data types are derived. This field indicates the number of other entities dependent on this point. For example, if the linkage field shows 2, then this means that two other entities (such as lines, arcs, or strings) are defined using this point number.
- **Annotation** – Each point may be annotated with three attributes: number, code, and height. These display on the screen and on the plot, if activated.
- **Annotation Settings** – The annotation elements display on the screen when the relevant annotation check box is selected on the *Multiple Point Edit* dialog box or the **Hot Button** options.
- **Attributes** – Press to add an attribute to the point.
- **Notes** – Press to add a note (optional).
- **Symbols** – Press to add a symbol from the symbols library.
- **Images** – Press to attach an image to the point.
- **Set Properties** – Press to set new properties for the point.
- **Get Properties** – Press to get new properties for the point.
- **Delete** – Press to delete a point.
- **Apply** – Press to apply new parameters for the point.
- **OK** – Press to save and exit the screen.
- **Cancel** – Press to quit the routine.

Defining a Line

A line is a connection between two points and is defined in the database by a start point and an end point. The bearing and distance of the line is totally dependent on the coordinate positions of these two points. The two points may have varying attributes. Lines can be computed, manipulated, edited, and deleted using a range of routines in MAGNET Office (Figure 4-2 on page 4-5).

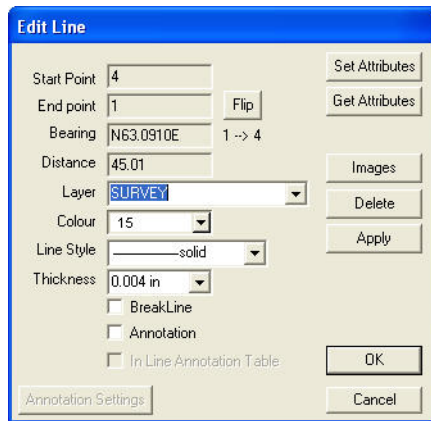


Figure 4-2. Line Attributes and Properties

Lines have the following properties:

- **Color** – Every line is assigned a color from the color palette.
- **Layer** – Each line resides on one layer.
- **Line Style** – This defines the appearance of each line on the screen and when plotted. The default style is solid. You can choose styles from a library of standard and user-defined styles.
- **Line Thickness** – This defines the width of the line in millimeters when plotted.
- **Breakline** – A line that joins two contourable points with heights. MAGNET Office treats a breakline as a line or arc across which you cannot form a triangle.
- **Annotation** – Each line may be annotated with two attributes, bearing and distance. These are displayed on the screen and on the plot if activated.
- **In Line Annotation Table** – Displays the bearing and distance of a line in the Line Annotation Table. If this table is plotted, the line is marked with a number, which is listed in the table with the appropriate bearing and distance.

Defining Arcs and Circles

An arc is defined as a start point, end point, and a center point or three points on its circumference. Circles are defined by a center point and a radius. Arcs and circles can be computed, manipulated, edited, and deleted using a range of routines in MAGNET Office (Figure 4-3).

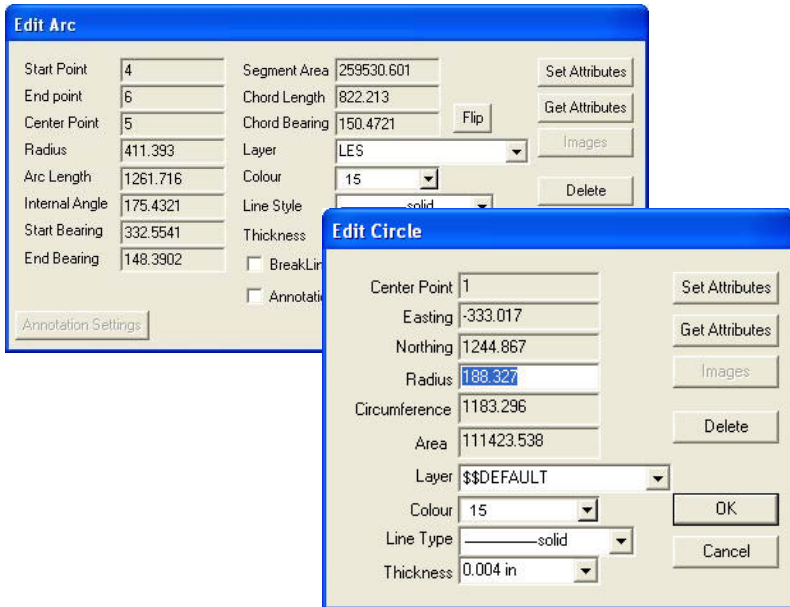


Figure 4-3. Arc and Circle Attributes and Properties

Arcs have the following properties:

- **Color** – Every arc and circle is assigned a color from the color palette.
- **Layer** – Each arc and circle resides on one layer.
- **Line Style** – Defines the appearance of each arc and circle on the screen and when plotted. The default style is solid. You can choose styles from a library of standard and user-defined styles.
- **Line Thickness** – Defines the width of the string in millimeters or inches, depending on the preferences you set on the **Unit Settings** tab of the **Project Settings** dialog box.

- **Annotation** – Each arc may be annotated with up to five attributes: radius, arc length, arc angle at the center point, bearing, and length of the chord. They will display on the screen and on the plot if activated. A circle cannot be annotated.
- **In Arc Annotation Table** – Displays the arc attributes in the Arc Annotation Table. If this table is plotted, the arc is marked with a number, which is listed in the table with the appropriate attributes.
- **Breakline** – An arc is defined by contourable points with elevations. MAGNET Office treats a breakline as a line or arc across which you cannot form a triangle. When a breakline is inserted along an arc, an arc to chord is set to define the number of triangles required along the arc.

Defining a String

A string consists of several points in a sequence joined by lines and/or arcs to become one entity. The string is a 3D string if all of the points that define it have elevation values. A string may have a smoothing algorithm applied to it for plotting purposes. Strings can be computed, manipulated, edited, and deleted using a range of routines in MAGNET Office (Figure 4-4).

Figure 4-4. String Attributes and Properties

Strings have the following properties:

- **Code** – A string may be labeled with a user-defined code. This code is necessary for some routines.

- **Color** – Every string is assigned a color from the color palette.
- **Layer** – Each string resides on one layer.
- **Line Style** – This defines the appearance of each string on the screen and when plotted. The default style is solid. You can choose styles from a library of standard and user-defined styles.
- **Line Thickness** – This defines the width of the string in millimeters or inches, depending on the preferences you set on the *Unit Settings* tab of the *Project Settings* dialog box.
- **Breakline** – A string may be defined as a breakline; however, all of the string point nodes should have elevations and be set as contourable. MAGNET Office treats a breakline as a line or arc across which you cannot form a triangle.
- **Smoothed** – A string may be smoothed with a spline algorithm. The smoothed format of the string is only used for plotting purposes and not for any form of computation.
- **Service** – A string may be a service for use in obstruction definitions and other routines.

Defining a Polygon

A polygon is a closed shape and consists of several points in a sequence joined by lines and/or arcs to become an entity. It may be selected by its centroid point. You can move the nodes of a polygon to change its shape. A polygon may be filled with a pattern, hatching, or shading. A polygon is the base entity for a lot, boundary, or pad. Polygons can be computed, manipulated, edited, and deleted using a range of routines in MAGNET Office (Figure 4-5 on page 4-9).

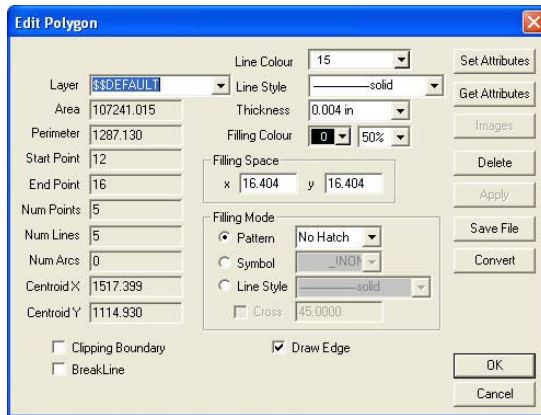


Figure 4-5. Polygon Attributes and Properties

Polygons have the following properties:

- **Color** – Every polygon is assigned a color from the color palette. This color is used for the lines around the edge of the polygon and for patterning and hatching if the polygon is filled.
- **Layer** – Each polygon resides on one layer.
- **Line Style** – This defines the appearance of the polygon edge on the screen and when plotted. The default style is solid. You can choose styles from a library of standard and user-defined styles.
- **Line Thickness** – This defines the width of the string in millimeters or inches, depending on the preferences you set on the *Unit Settings* tab of the *Project Settings* dialog box.
- **Filling Color** – A separate color may be applied to the fill, with a color percentage applied to the shading. The shading is set in the *Filling Color* drop-down menu.
- **Filling Space** – The X and Y values control the spacing between lines, used for hatching and spacing between symbols.
- **Filling Mode** – There are three options available:
 - **Pattern**: The drop-down menu offers a range of pattern styles, including the shading selection and shading percentage.

- Symbol: A fill pattern made from a repeated MAGNET Office symbol.
- Line Style: A fill pattern made from parallel lines at the filling space offset using the line style selected. If the **Cross** check box is selected, the pattern becomes a cross-hatching of lines using two sets of crossing parallel lines with the style and specified angle of crossing.
- **Clipping Boundary** – A polygon may be used to exclude contours within the polygon boundary.
- **Draw Edge** – The user may choose to draw an edge around the polygon or leave the polygon without an edge. The latter is useful where the polygon is created from existing lines, or the polygon is used for a clipping boundary, but the edge is not shown.

Creating Text

Text is a group of alpha, numeric, or alphanumeric characters. Text within MAGNET Office can be one word, one number, or a mix of words and numbers complete with spaces to comprise a sentence. Text is displayed on one or multiple lines (Figure 4-6 on page 4-11).

Text within MAGNET Office is non-scalable, so if text is created with a height of 3.5 mm, the text will always plot at 3.5 mm regardless of the plotting scale. There are several ways to create and store text within MAGNET Office.

For example, the text for a street name may be stored at a given position and orientation. This is created using the ***Insert Text*** command, where the user enters the street name.

The text for bearing and distance may be stored in accordance to a line, so its orientation and location depend on the length and direction of the line at the time the text is inserted. This is created using **Insert Bearing and Distance** with the value of the line bearing and distance extracted from the database and displayed as text. This is called **Specific Text**.

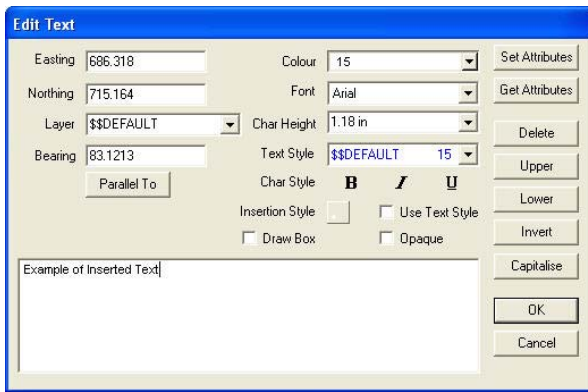


Figure 4-6. Text Attributes and Properties

Text has the following properties:

- **Text String** – Characters comprise a text string. Text composition may be changed. A text string can be a single line of text or multiple lines of text held as one text unit. Text may be imported from a file.
- **Text Style** – The text style defines the appearance of the text. It allocates a font, height, color, and optional character style to the text; however, you may edit these styles individually. Using a text style speeds up text entry and controls presentation standards.
- **Insertion Point** – When text is created, a coordinate location or position is recorded in the database for this text. This position is usually selected by the user with the mouse prior to text entry or the position the text may be moved to. The insertion position relative to the text string is controlled by the Insertion Style.
- **Insertion Style** – The insertion style specifies where the insertion position is located relative to the text string. There are nine text insertion styles, from the lower left corner of the text through the center of the text to the upper right corner of the text.
- **Layer** – Each text string resides on one layer.
- **Bearing or Angle** – The bearing or angle that defines the orientation of the text in the X-Y plane. Bearings are measured clockwise from north (zero degrees) in the same manner as lines.

Angles are measured counter clockwise from the X-axis or east (zero degrees). Access this setting by clicking **Settings ▶ Project Settings ▶ Unit Settings ▶ Rotation By**.

Specific Text Certain text strings are created as specific text. This text is created when MAGNET Office derives the text string from information in the database using commands on the **Insert Text** submenu. This text uses the text style and other settings used for annotation.

See also “Defining Annotation” on page 4-12.

Defining Annotation Annotation is an attribute of points, lines, arcs, and other entities. When the annotation of an entity is activated, relevant information is displayed beside the entity. The format is taken from the current annotation settings set in the project.

The annotation is automatically updated if the entity is edited or moved. For example, if the height of a point is modified, then the annotation of the height is updated to reflect the new elevation. If a line is moved, the annotation of the bearing and distance is updated.

If the height, point, or bearing and distance of a line was created as specific text, the text is not updated when the point or line is modified.

See also “Specific Text” above.

Defining an Arrow

An arrow is a special line or series of lines with an arrow head at one end. The arrow can be used as an indicator on a plan. Various arrow head styles are available with user-defined dimensions for the arrow head. Arrow heads are non-scalable, and they will always plot at the specified size in millimeters regardless of the plotting scale (Figure 4-7 on page 4-13).

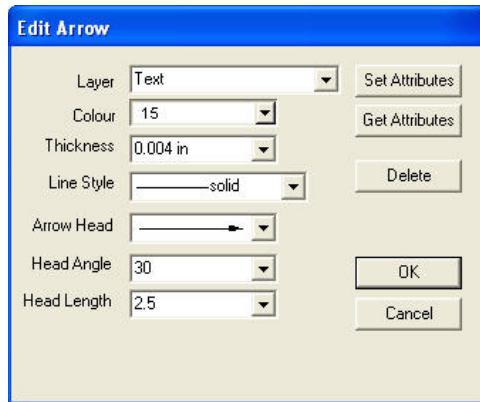


Figure 4-7. Arrow Attributes and Properties

Edit Arrow has the following properties:

- **Color** – Every arrow is assigned a color from the color palette.
- **Layer** – Each arrow resides on one layer.
- **Line Thickness** – Defines the width of the string in millimeters or inches, depending on the preferences you set on the *Unit Settings* tab of the *Project Settings* dialog box.??
- **Arrow Head** – This may be selected from several available types.
- **Arrow Angle** – The arrow angle is measured at the pointed end of the arrow head. The maximum angle is 90 degrees.
- **Arrow Length** – The arrow length is the length of the arrow head measured in millimeters on the plan.

Defining Dimensions

The dimension entity measures the distance between two selected points and writes the dimension as an annotation above or below a line with arrows at either end. Any of these attributes may be edited (Figure 4-8 on page 4-14).

The arrows used for dimensions are defined as arrows.

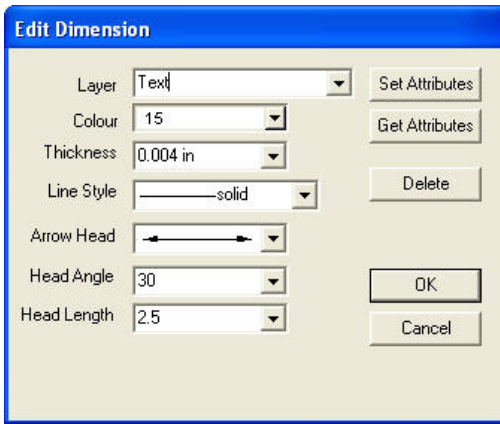


Figure 4-8. Dimension Attributes and Properties

Edit Dimension has the following properties:

- **Color** – Every dimension is assigned a color from the color palette.
- **Layer** – Each dimension resides on one layer.
- **Line Thickness** — This defines the width of the string in millimeters or inches, depending on the preferences you set on the *Unit Settings* tab of the *Project Settings* dialog box.

Derived Data Types or Entities

Several other entities are complex or derived entities. They are created from a combination of the basic data types and they include the following:

Creating Boundaries

Click **Surface ▶ Create Boundary** to create a boundary. A boundary is a series of interconnected lines and/or arcs, which form a closed figure. A boundary is a named polygon to identify it for use in a variety of routines (Figure 4-9 on page 4-15).

Figure 4-9. Boundary Attributes and Properties

You can use a boundary to do the following:

- Compute volumes within the boundary
- Delete triangles inside or outside the boundary
- Select entities inside the boundary for other options, such as editing

Defining Lots (Allotments)

A lot is a closed figure made up of line and/or arc segments. A lot is a particular type of polygon with several additional attributes to define it as an allotment in a subdivision. Lots are created using the **Survey ► Subdivision** command (Figure 4-10 on page 4-16).

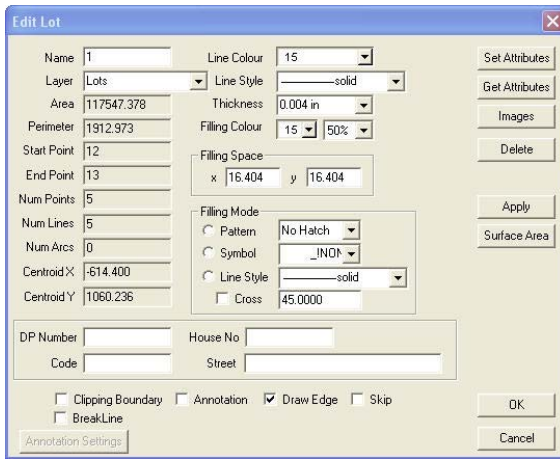


Figure 4-10. Lot Attributes and Properties

The Lot database helps designers maintain lot details that can be printed. Lots can be automatically annotated with bearing, distance, area, lot numbers, and angles as required for the final plan presentation.

Lots have the following attributes and properties:

- **Boundary Points** – The set of point numbers defining the corners of each lot. MAGNET Office stores the details of the lines or arcs joining the points. These lines and arcs can be annotated.
- **Lot Number** – A unique number or letter identifier for each lot. Lot numbers can be annotated.
- **Area** – The area contained within the lot. The area can be annotated at the lot center.
- **Code** – Lots may be allocated to separate groups defined by codes. This is useful in staged subdivisions where different groups of lots may need to be manipulated individually. The code may also be used for other types of grouping.
- **DP Number** – An identifier representing the lot's DP number that can be optionally stored for the lot.

- **House Number and Street Name** – Optional identifiers representing the lot's house number and street name that can be stored for the lot.
- **Annotation** – A lot may be annotated with a number of items. These are defined in the **Lot Annotation Settings**. The annotation elements are only displayed on the screen when the relevant annotation check box is selected from the Hot Button options.

Creating Pads

Click **Surface ▶ Create Pad** to create pads. A pad is a series of interconnected lines and/or arcs that form a closed figure around the extents of the building or other pad. A pad is a named polygon to identify it for use in the design of a pad and side slopes suitable for the placement of buildings (Figure 4-11).

The 'Edit Platform' dialog box contains the following fields and options:

- Name:** Flat 1
- Layer:** Base1
- Area:** 194970.276
- Perimeter:** 2577.293
- Start Point:** 3
- End Point:** 1
- Num Points:** 3
- Num Lines:** 3
- Num Arcs:** 0
- Centroid X:** -59.861
- Centroid Y:** 1309.085
- Line Colour:** 15
- Line Style:** solid
- Thickness:** 0.004 in
- Filling Colour:** 0, 50%
- Filling Space:** x: 16.404, y: 16.404
- Filling Mode:**
 - Pattern: No Hatch
 - Symbol: _IND_
 - Line Style: solid
 - Cross: 45.0000
- Clipping Boundary
- BreakLine
- Draw Edge

Buttons on the right side include: Set Attributes, Get Attributes, Images, Delete, Apply, Save File, Convert, OK, and Cancel.

Figure 4-11. Pad Attributes and Properties

Defining Structures

A structure is a series of interconnected lines and/or arcs that form a closed figure around the extents of a structure. Click **Survey ▶ Define Structure** to save structures to a separate file. Structures can be inserted into any job and then moved, located, and rotated using several tools. Finally, the structure is saved to an exact position, where the points are coordinated and fixed.

A structure may represent a house outline or other features such as a dam, a cul-de-sac, a traffic island, or a roundabout (Figure 4-12).



Figure 4-12. Structure Shapes and Options

Triangles in Digital Terrain Models (DTM)

The triangular mesh or triangle irregular network (TIN) is the basis of the digital terrain modeling options in MAGNET Office.

Each triangular mesh represents a surface, also known as a digital terrain model (DTM). The triangular mesh is used for a wide range of computations, including the calculation of contours, cross sections, and earthworks volumes.

A DTM is created using the **Surface ▶ Create DTM** command. All points are on active layers and are used to form the mesh. You can also create a DTM from preselected points.

Each triangle is defined from three points within the database. Triangles can be added, deleted, and edited (rearranged) by the user to give a more accurate terrain model. There are several functions for manipulating triangles on the DTM menu.

Triangles are displayed on the screen in the chosen color with a dot at the centroid of each triangle. The triangular mesh is displayed when the *Surface Model* check box is selected on the DTM Settings window.

You can create multiple surface models in one project, which are saved in the ccx file.

You can merge one triangular mesh into another to form a third surface. The merged data replaces the triangles in the original triangular mesh.

Interpolating Contours

Contours are interpolated directly from the current triangular mesh. Contours are displayed on the screen when the **Contours** field is selected on the **DTM Settings** dialog box.

The intervals, color, and line style for the major and minor contours are set when the DTM is created and may be edited from the **DTM Settings** dialog box.

Contours are usually displayed as straight lines in the project, although they may be set as smoothed from the **DTM Setting** dialog box. Contours can be plotted as a straight line or smoothed contours.

Defining Breaklines

MAGNET Office defines a breakline as a line or arc joining two contourable points, which a triangle cannot cross. The points must have heights and be at the triangle vertex in the triangular mesh. On the ground, a breakline represents a line of discontinuity in the surface grade of the line of a sharp grade change (Figure 4-13).

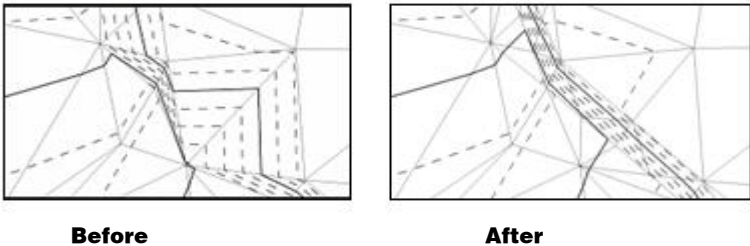


Figure 4-13. Triangles and Contours Before and After Inserting Breaklines

Examples of breaklines include:

- Top and bottom of a bank
- Invert of a drain
- Top and bottom of a curb
- Ridge of a hill

It is essential for a surveyor to code the survey points joining breaklines correctly in the field. The breaklines are defined when the

data is reduced and the lines are created from the stringing code. All valid breaklines must then be inserted into the triangular mesh or the surface model will be incorrect.

Whenever a line is created, the breakline flag for that line is set to **On** or **Off**. Before creating a DTM or inserting breaklines into an existing DTM, the surveyor should check for any breaklines that cross. Crossing breaklines will be ignored. You can check for crossing breaklines using the **Surface ▶ DTM Tools ▶ Check Breaklines** command. When breaklines are inserted into a DTM, additional triangles are formed to ensure that triangle edges lie along the breakline and no breakline crosses a triangle edge. Additional internal points are created to enable the formation of these new triangles. These points may not be selected or edited, but they may be selected when deleting and adding triangles.

If the triangular mesh does not accurately represent the ground surface, all data computed from these triangles will also be incorrect. This means that contours, cross sections, and volumes will be unreliable.

Defining Curb Strings

A curb is a particular type of string that is usually defined to represent the curb line through intersections between one road and another. The curbs are usually formed along the seal or bitumen edge (lip of the curb) or the invert of the curb. Curbs are defined with a natural surface profile extracted from a DTM or from the elevations that exist on the points used as nodes of the curb.

Curbs have individual names and can be viewed, edited, and plotted. If the design profile is modified, the new levels may be saved back to the Survey View to update the elevations on the curb's node points.

Curbs are frequently defined along all of the profile lines that require to be graded through an intersection. These curbs may be used in the Intersection Design View to complete the design levels and surface.

Designing Intersections

A number of Curb Strings may be incorporated into an Intersection Design. The intersection may be viewed and the levels on all of the curbs fine tuned to produce a suitable surface model through the intersection. Each individual curb may be modified to alter the surface model.

The finished surface model and the final levels on the various curbs may be saved back to the project data in the Survey View, so the node points on the curbs are updated with the finished elevation values.

Designing Alignments

An alignment is usually the center line of a road, pipeline, service, or other feature. It is often used to produce profile and cross section data and is essential for designing a road. An alignment consists of a number of intersection points (IP) joined together with lines. An IP is linked to a point in the database. If it represents a curve in a road, then the IP may have arc and spiral parameters defined. It is not necessary to create any line segments before creating an alignment, although this may be helpful. Existing line or string data may be converted to an alignment (Figure 4-14 on page 4-22).

Each road dataset in the Roads View is named by its alignment name.

When an alignment entity represents the horizontal geometry of a road, it contains all the road details, and if detected, the alignment will also erase the road data.

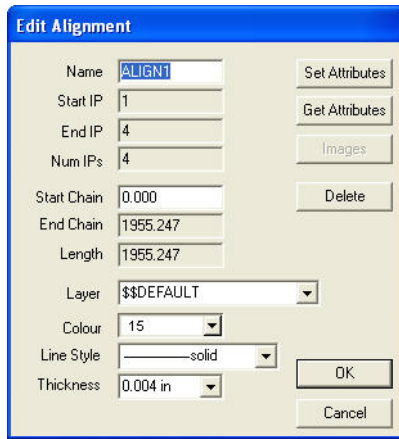


Figure 4-14. Alignment Attributes and Parameters

Alignments have the following attributes and parameters:

- **Name** – Each alignment has a unique name. Do not include spaces in the name.
- **Layer** – Each alignment resides on one layer.
- **Color** – Each alignment is assigned a color from the color palette.
- **Line Style** – Defines the appearance of an alignment on the screen and when plotted.
- **Line Thickness** – Defines the width of the string in millimeters or inches, depending on the preferences you set on the **Unit Settings** tab of the *Project Settings* dialog box.
- **Start IP** – The point number of the first IP on the alignment.
- **End IP** – The point number of the last IP on the alignment.
- **Num IPs** – The number of IPs in the alignment.
- **Start Chainage** – The nominated running distance at the start IP of the alignment. This defaults to 0.000 but may be set to another value.
- **End Chainage** – The running distance at the end IP of the alignment.

- **Length** – The length of the alignment in meters from the first to the last IP.

Viewing DTM in 3D

The MAGNET Office 3D Viewer displays a 3D visualization of all digital terrain models that you can view from any selected position. The 3D View is exported to the w3D file from the **File ▶ Export** menu in the Survey View. The current triangular mesh is used to define the 3D View, together with lines and alignments on active layers.

If alignment is embedded in the w3D file, then you can view the surface by moving along the road defined by the alignment, as if driving along it.

Additional Data Types

MAGNET Office has additional data types, which include: symbols, line styles, title blocks, layers, angular units, and distance units.

Defining Symbols

A symbol is a graphical representation of a point feature. Each point in the MAGNET Office database may have a symbol attached to it using the Symbol ID. The simplest symbol is a cross or circle, which marks the point position; however, symbols may also be complex, such as a symbol for a special type of tree.

Examples of symbols include:

- Trees
- Survey Marks
- Point crosses and circles (example: point markers)
- Sewer manholes

Some symbols are simply a diagrammatic representation of a point (example: survey mark) at a specified size on the paper. Other symbols provide a true representation of the size, shape, and

orientation of the point. You can create these symbols so they are scalable and rotatable, ensuring complete flexibility of point representation. Symbols can contain areas of fill or text and can be saved so the lines joining the symbol will clip at the symbol boundary.

In Survey View, click **Library** ▶ **Symbols** to access symbol options. Once a symbol is used in a project, it is embedded in the project file `cc6`, and is made available to other operators who may not have the same library as the original user (Figure 4-15).

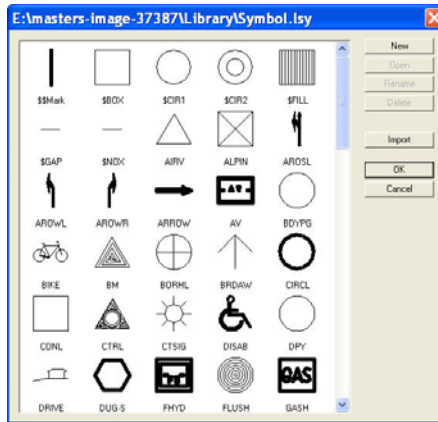


Figure 4-15. Symbol Library

Defining Line Styles

A line style is a graphical representation of a line feature, equivalent to a symbol repeated along the length of a line. Line styles may be standard styles, such as a solid or dashed line, or user-defined styles, such as a fence line.

Line style representations include:

- Gas main
- Water pipe
- Drainage line
- Vegetation outline

- Power lines

Defining Title Blocks

A title block contains the layout for the plan when it is plotted. Users cannot plot a plan from the Survey View, a profile or cross section from the Roads View, or a profile from the Drainage or Sewer Views without nominating a title block.

A title block can be a simple rectangular border of four lines or a complex layout containing text, lines, and symbols with areas for information about the company and the project being plotted. It may also contain keywords that are populated by the individual labels applied to each project in the Library menu under the Title Block option.

The title block consists of two specific areas:

- The drawing area or extents of the plot window: This area contains the project data. This area is displayed when a plot window is created.
- The title area outside the plot window: This area is used for standard details, such as company information, plan titles, approvals, resolutions, and so forth. A symbol legend may be included. The plan scale and other keywords are also included in this area.

Several title blocks are designed to represent the different sizes of plans (example: A1, A2, A3, or A4 sizes) that are commonly used. Each title block has its own specific layout and detail in the title area.

The title block is designed in millimeters. The drawing area is empty with the entities needed for the title detail outside this area. You can design several title blocks in one MAGNET Office project and save them using the Title Block command in the Library menu. The title block file is named and saved in the Library folder. It has the extension *.tbk*.

You can include special keywords in the title details to represent certain project data. The keywords are replaced by this data during

the creation of the Linked Drawing. You can add other title details in the Drawings View prior to final plotting.

Defining Layers

Layers enable you to edit, manipulate, and present data with greater speed, accuracy, and flexibility than normally possible. With layers, you can view, edit, and plot related aspects of the model either independently or in any combination.

Think of a layer as a clear sheet of film on which all related items are drawn. You can view this sheet separately or combined with several other sheets to produce a view of an area. For example, in a topographic survey, data can be placed in layers according to whether the data represents a road, fence, natural surface, property boundary, and so forth.

The current layer is always displayed on the **Properties Toolbar** and is labelled with an asterisk on the *Layer Settings* screen. In most cases where entities are created, they are placed on the current layer. The two exceptions to this rule are:

- Feature coded raw survey data: The data is placed in predefined layers during data reduction in the Neutral File View or when imported from an ASCII file. Any points with feature codes are placed on the layer specified for the feature code in the Survey Codes library.
- Various routines allow you to specify which layer is used for the data at the time of computation. You can define a layer in the data entry screen, convert a contour to a string, create side slope points in pad design, and save design data from the Roads View to the Survey View.

Configuring Angular Units Entry

MAGNET Office can accept different angular units. Angles and bearings are entered in degrees or grads, and projects can be configured for degrees or grads. The default setting for all projects is shown on the *Unit Settings* tab of the **Settings ▶ Project Settings** menu in the Survey View.

Angles and bearing entered in degrees are entered in the format DDD.MMSS. For example: 270 degrees, 25 minutes, 30 seconds is entered as 270.2530; decimal degrees are not required.

Gons are entered as required, such as *300.4722*.

Defining Bearings and Angles

These terms have different interpretations in the surveying and engineering professions to describe different methods of recoding horizontal angles and directions. The definitions for MAGNET Office are as follows:

- **Angle** – A horizontal or vertical angular measurement with no particular reference to a known azimuth. An example is the difference between a backsight and foresight observation.
- **Bearing** – The clockwise angle turned from north. In some countries, this is referred to as a direction. When referring to a bearing in MAGNET Office, north azimuth is always assumed.
- **Quadrant Bearings** – These relate to the four points of the compass in degrees or gons.
 - Bearings from North: DDD.MMSS for DDD degrees, MM minutes, SS seconds
 - Degrees, Quadrant Bearings (Input):
 - QDD.MMSS for DD.MMSS degrees in the Q quadrant
 - XDD.MMSSY for DD.MMSS from compass point X to Y
e.g. N45.3320E
 - DD.MMSSXY as above, e.g. 45.3320NE
 - XYDD.MMSS as above, e.g. NE45.3320
 - Degrees, Quadrant Bearings (Output):
 - XDD.MMSSY for DD.MMSS from compass point X to Y, for example, N45.3320E
- **Swing** – This is used to define an angle to adjust the azimuth of a survey. Depending upon the menu command being used, swings can be applied to individual observations, all observations from one occupied station, or to all observations in a traverse.

Defining Distance Units Entry

Distances are units that can represent meters, decimal feet, links, millimeters, etc. Each project is created so the distance unit is appropriate to the work being done.

The user has the option in some routines to mix the input definition of distances. The working distance unit can be meters, but data can be entered from existing plans in feet or links with the automatic conversion to meters prior to the computation of the point positions.

Displaying Scale Factors Each project can have a user-specified scale factor. This is set and will display on the **Unit Settings** tab of the **Settings ▶ Project Settings** menu in the Survey View. Set this value for the project before any data is entered.

The scale factor is applied to the ground distance in all computations of entities, such as points, lines, arcs, and areas. The resulting projection distance is used to compute the point coordinates. The scale factor is also applied to convert the projection distance to the ground distance when data is displayed from the coordinate values.

If all of the work for a project was computed in error with no scale factor applied, you can rectify this using the **Survey ▶ Translate/Rotate** menu in the Survey View. Use this command to apply the scale factor to all points radiated from one known point.

Survey View: File Menu

The Survey View displays upon first opening MAGNET Office. By default, all toolbars and rulers are opened. This section describes the different Survey View menu commands (Figure 5-1) on the **File** menu located on the main menu bar

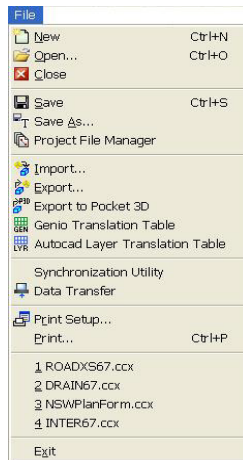


Figure 5-1. Survey View – File Menu

You can move between the Survey View, the Drawing View, the Cross Section Editor, and the Raw Data Editor by clicking the appropriate icon on the **Modules** toolbar (Figure 5-2).

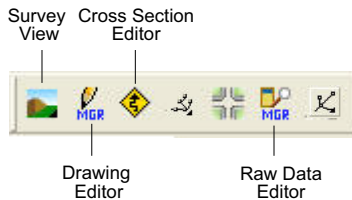


Figure 5-2. Modules Toolbar

This chapter and subsequent chapters (6-13) describe the following Survey View menu commands located on the main menu bar:

- “Survey View: Edit Menu” on page 6-1
- “Survey View: View Menu” on page 7-1
- “Survey View: Settings Menu” on page 8-1
- “Survey View: Insert Menu” on page 9-1
- “Survey View: Modify Menu” on page 10-1
- “Survey View: Inquiry Menu” on page 11-1
- “Survey View: Survey Menu” on page 12-1
- “Survey View: Design Menu” on page 13-1
- “Survey View: Surface Menu” on page 14-1
- “Survey View: Tools Menu” on page 15-1
- “Survey View: Library Menu” on page 16-1
- “Roads View Editor” on page 18-1

About the Survey View

When a new project is created or an existing project is opened, the Survey View screen displays (Figure 5-3 on page 5-3). To identify the Survey View tools, see Table 5-1 on page 5-3.

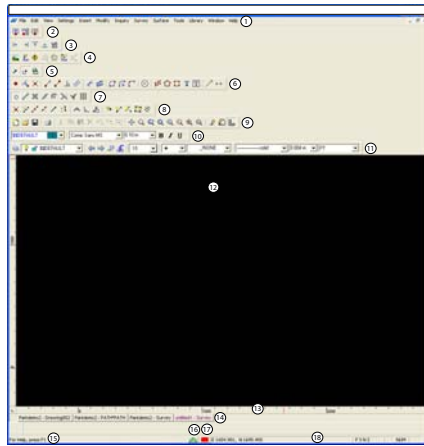


Figure 5-3. Survey View

Table 5-1. Survey View Window Description

Item Number	Item Description
1	Menu Bar
2	Point Information Toolbar
3	Align Toolbar
4	Modules Toolbar
5	Input Modes Toolbar
6	Add Entity Toolbar
7	Snap Modes
8	Survey Tools Toolbar
9	Standard Toolbar
10	Text Toolbar
11	Properties Toolbar
12	Project Area
13	Ruler
14	Project Tabs
15	Prompt Line
16	DTM Button
17	General Settings (Hot Button)

Table 5-1. Survey View Window Description (Continued)

Item Number	Item Description
18	Status Bar

File Menu Options

This section describes the different Survey View menu commands on the **File** menu located on the main toolbar.


In This Section:

- “New” on page 5-5
- “Open” on page 5-7
- “Close” on page 5-8
- “Save” on page 5-9
- “Save a Project File to Another Name” on page 5-10
- “Project File Manager” on page 5-10
- “Import” on page 5-12
- “Export” on page 5-19
- “Export to Pocket-3D” on page 5-25
- “Genio Translation Table” on page 5-26
- “Autocad Layer Translation Table” on page 5-29
- “Synchronization Utility” on page 5-31
- “Data Transfer” on page 5-36
- “Print Setup” on page 5-37
- “Print” on page 5-38
- “Exit” on page 5-39

New

Each newly created project also creates a new database to store data for that project. Projects are given a name and may be opened and edited.

To create a new project, do one of the following:

- Click **File ▶ New**.
- Press the shortcut keys **Ctrl+N**.
- Press **Create a New Project** when the **Startup Wizard** appears upon first opening MAGNET Office.
- Click the **New**  icon on the **Standard** toolbar.

The **Startup Wizard** appears.

1. In the **Project Details** dialog box (Figure 5-4), type project information in the empty fields, and then press **Next**.

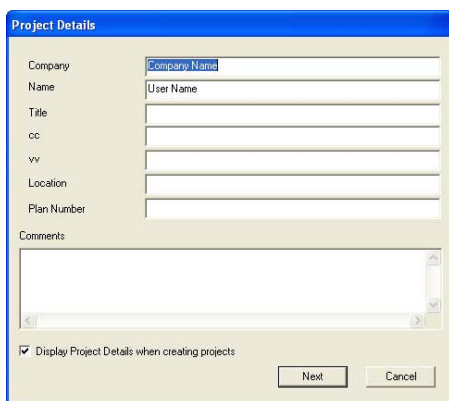


Figure 5-4. Project Details

2. On the **Unit Settings** dialog box (Figure 5-5 on page 5-6), modify the unit settings as needed, and then press **Finish**.

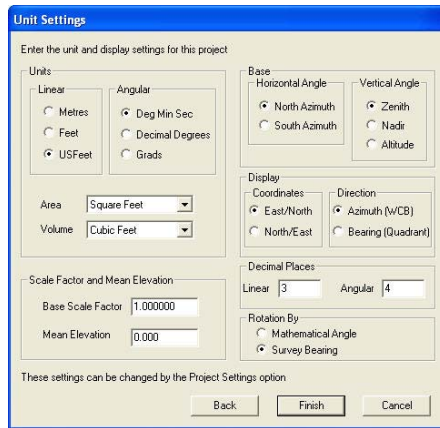


Figure 5-5. Unit Settings

3. On the *Create a New Project* dialog box (Figure 5-6), choose to create a new project by importing or entering data, or create a project with no data.

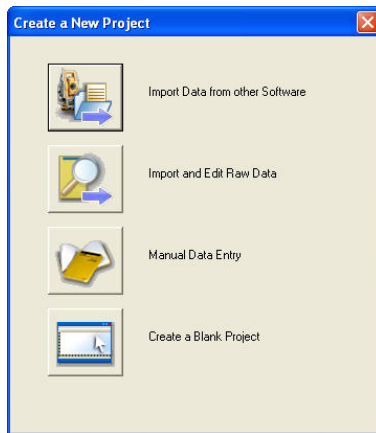


Figure 5-6. Create a Blank Project


Different dialog boxes display, depending on the option you selected. The new project dialog box may not fill the maximum space of the screen.

To ensure all windows are maximized when opened:

- Click **Settings** ▶ **Program Settings** ▶ **Preferences**.
- Check mark the **Maximize when opened** box, then press **OK**.

Open

To open an existing project, do one of the following, do the following:

- Click **File** ▶ **Open**
- Press the shortcut keys **Ctrl+O**
- Click the **Open**  icon on the **Standard** toolbar.
The **Open** dialog box appears (Figure 5-7).

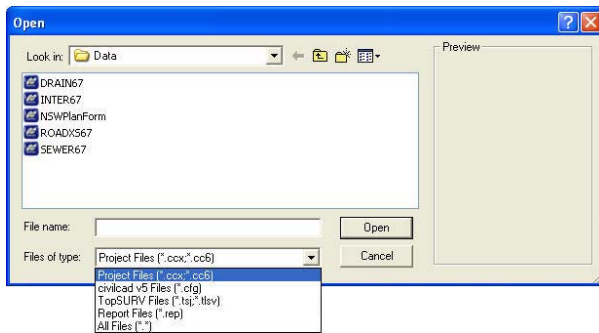


Figure 5-7. Open Dialog Box

Search for the file you want, and press **Open**. The project data displays on the screen (Figure 5-8 on page 5-8).

Alternatively, click on the **File** menu and select one of the most recently used files displayed at the bottom of the menu.



Figure 5-8. File Menu – Open Recent File

There may be one or more files opened at the same time. If you select a file from the **File** menu and nothing happens, that file may already be opened.

To move between open files, do the following:

- Click on the project tabs beneath the display area
- Click on the **Window** menu (Figure 5-9), and select the file you want to view.

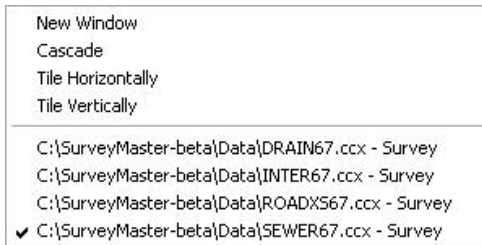



Figure 5-9. Window Menu

Close

You can close a project file without exiting MAGNET Office.

To close a project file:

1. Do one of the following:
 - Click **File** ► **Close**.

- Click the **Close**  icon on the upper-right corner of the window.
2. If the project is an existing file and has been altered, then a confirmation message appears asking if you want to save your changes. Press **Yes**.

If the project is new and you have not given it a name, then the **Save As** dialog box appears. Type a file name, select a folder in which to save the file, and press **Save**.

Save

When a new survey window appears, it is automatically given a temporary title, untitled1-Survey. Subsequent windows are titled untitled2-Survey, untitled3-Survey, and so on.

To save a project file:

1. Do one of the following:
 - Press the shortcut keys **Ctrl+S**.
 - Click **File** ► **Save**.

If the project is new and doesn't have a name yet, then the **Save As** dialog box appears (Figure 5-10).



Figure 5-10. Save As Dialog Box

2. Type a file name, and select a folder in which to save the file.
3. Select an appropriate file type from the **Save as type** menu.
4. Press **Save**.

Save a Project File to Another Name

To change the name of a project file and save it:

1. Open the file if it is not already opened.
2. Click **File ► Save As**.

The **Save As** window appears (Figure 5-11).

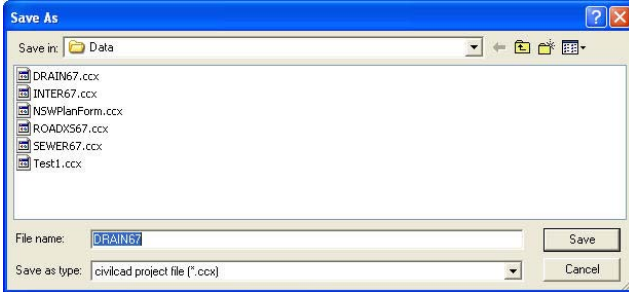


Figure 5-11. Save As Window

3. Type a new name in the **File Name** field.
4. Press **Save**.

Project File Manager

All data in MAGNET Office is stored in a single project file with a **.ccx** extension. The project file is self-contained and space efficient. The Project File Manager provides a facility to store any files related to a project into a single file, making file management tasks easier. You can also attach other files (except other project files) relating to the project.

To manage project files, do the following:

1. Click **File ► Project File Manager**.

The *Project File Manager* window appears (Figure 5-12 on page 5-11).

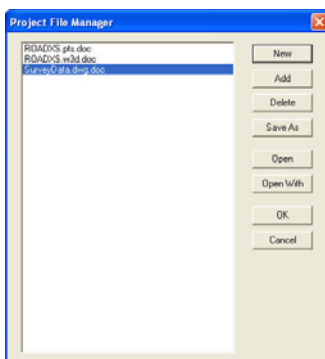


Figure 5-12. Project File Manager Window

- To create a new Word or Excel document and add it to the project, then press **New**.

The *New Document* dialog box appears (Figure 5-13).

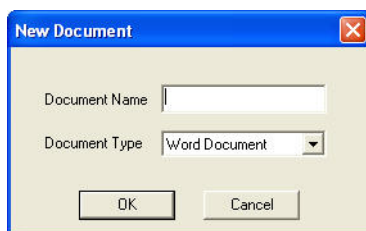


Figure 5-13. New Document Window

- Make the necessary modifications. For button descriptions, see Table 5-2.
- Press **OK** to save your changes.

Table 5-2. Project File Manager Button Descriptions

Button	Description
New	Creates a new Word or Excel document and adds it to the project.
Add	Adds a selected file to the project.
Delete	Deletes the selected file from the project.
Save As	Saves a copy of the selected document from the project outside of MAGNET Office. This button does not delete the file from the project.
Open	Opens the selected file with an external application.

Table 5-2. Project File Manager Button Descriptions (Continued)

Button	Description
Open With	Opens the selected file with an external application of your choice.
OK	Saves and exits the document file.
Cancel	Quits the routine.

Import

You can import data into a MAGNET Office project in several ways, including importing files created by other software.

The file formats listed in Table 5-3 are compatible with MAGNET Office.

Table 5-3. File Formats (Read)

Compatible MAGNET Office File Formats	
civilcad 5 ASCII files	Genio files (*.gen / *.crd)
MicroStation Files (*.dgn)	Carlson Road (*.rd5)
Autocad Drawing files (* .dwg & * .dxf)	TopSurv files (*.tsj / *.tlsv)
Autocad files (*.dxf)	TopSURV CTF (*.ctf)
ASCII Point (*.pts / *.asc / *.csv / *.xyz / *.txt)	LandXML files (*.xml)
Topcon files	Hayes Data (*.dat)
Microstation (*.dgn)	3DMC Project (*.tp3)
Standard Survey Roads (*.hal / *.val. / *.rd)	3DMC (*.rde. / *.tn3/* .pt. / *.ln3)
CLIP Roads (*.plt / *.alz / *.trv)	ISPOL Road (*.ali / *.ras / *.sc1)

Raw data from instruments can also be imported using this routine. The following file formats are supported in MAGNET Office (Table 5-4 on page 5-13):

Table 5-4. Raw Data Instruments Supported by MAGNET Office

Compatible MAGNET Office File Formats	
Topcon GTS-700 (*.GT7 /*.GTS7)	Topcon CivilSURV (*.GT7)
Leica GSI (*.GSI)	Nikon AP800 (*.AP8 / *.RAW)
Sokkia SDR33 (*.SDR)	Trimble and Geodat 5600 (*.JOB *.ARE)
Wild GRE4 (*.GRE)	



In most cases, it is advisable to import data into a new project and then copy and paste it into an existing project.

To create a new project file, see “New” on page 5-5.

To Import a File, do the following:

1. Click **File ► Import**. The *Open* dialog box appears (Figure 5-14).

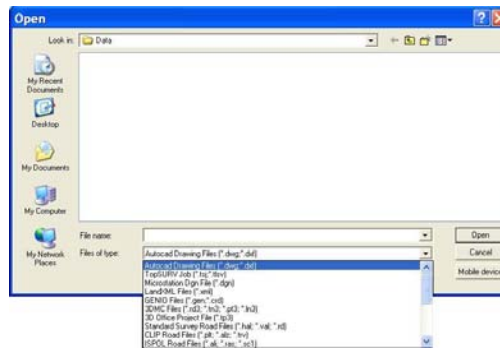


Figure 5-14. Import-Open

2. Select the appropriate file type from the **Files of type** drop-down menu.
3. Press **Open** (Figure 5-14).



Imported data may not display as it appears in other applications because data structures are often different. MAGNET Office can read the data from a range of formats, however, some of these formats may not be fully supported. For example, Autocad.dwg files may have complex entities compressed into blocks. These blocks would need to be exploded in Autocad prior to reading the data in MAGNET Office.



Some files contain embedded point numbers. If the current project contains existing data, then you may prefer to import these files into a new, empty project and then copy and paste the data into the open project so the points are renumbered.

Some import options may trigger the software to prompt for additional settings in order to be confirmed.

When you import an AutoCAD file, the *Convert AutoCAD* dialog box appears. The following parameters are set:

- Distance Unit Conversion –
 - No Conversion: enable to import the geometry into the units specified in the project settings, units.
 - From Inches: enable this option when you are inserting a drawing that was prepared in architectural units. Distances will be multiplied by 12 to convert to Feet. If the units are set to metric, the dialog will read From Millimeters in the Distance Unit Conversion area.
- Point Creation –
 - Create points for 2D polylines: check mark this box to automatically create points on 2D polylines during import. To reduce import time on large drawings, this option should be turned OFF and the **Create Points from Entities**

command in the **Modify** menu used instead to create only the required points for the project.

- Set points with elevation contourable: check mark this box to set the contourable flag to ON for any point with elevation.

A point must be set to contourable to be used in surface creation, even if it has a defined elevation.

- Blocks – check mark the Explode Checked Blocks box to explode all checked blocks on import.

Those blocks left unchecked in the box below will be inserted as symbols. If the symbols are not present in the project, they will be automatically created and added to the project symbol library.



Figure 5-15. Import an AutoCAD File

AutoCAD Before importing the data, decide whether the data is to be treated as ground or plan units. Point creation along 2D polylines is optional (Figure 5-15).

Microstation In the *Convert DGN* dialog box, check mark the *Create points for 2D polylines* box. This allows you to decide whether points can be generated for 2D Polylines (those with no height) or not (Figure 5-16 on page 5-16).



The TopSURV utility must be installed to run this feature.

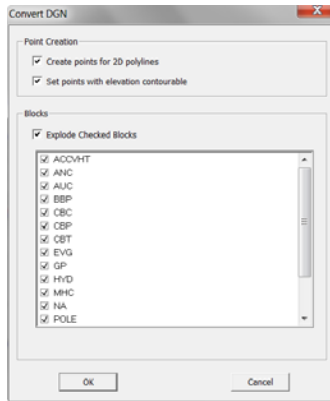


Figure 5-16. Import a Microstation File

Genio file contain points with a specific coding convention. In the *Import Genio* dialog box, you must specify how data is to be coded. Select either Two character or Four characters (Figure 5-17).

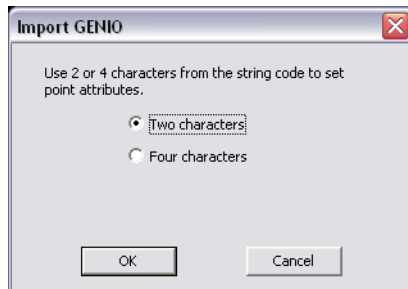


Figure 5-17. Import a Genio File

3DMC selected components of 3DMC files can be imported. From the *3DMC Import Files* dialog box, check mark any or all of the following import files: Alignment, Points, Linework, or DTM (Figure 5-18 on page 5-17).

From the *Import 3DMC* dialog box, enter an interval for the Cross Section (Figure 5-18 on page 5-17).

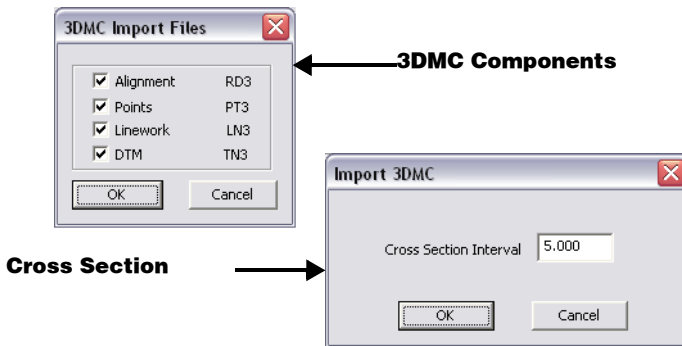


Figure 5-18. Select the Desired 3DMC Components

3DMC Project all components for 3DMC project files will be imported (Figure 5-19) On the *Import 3DMC* dialog box, enter an interval for the Cross Section.

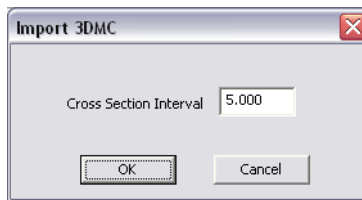


Figure 5-19. Enter an Interval for Cross Section for 3DMC Project Files

ASCII data is a text format listing point numbers, coordinates, and elevation of points. From the *ASCII File Format* dialog box, codes assigned for each point will allow each imported point to be imported reading the Survey codes and code settings Library. This will automatically set point properties and layers for each point; if string and command codes are used points will be strung together as coded (Figure 5-20 on page 5-18).

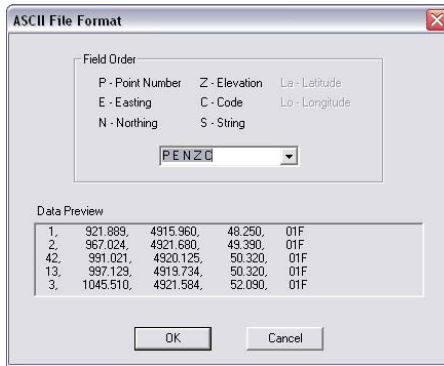


Figure 5-20. Automatically Set Point Properties and Layer for ASCII Files

Raw Files from survey instruments can be imported directly into the software. This may be more useful for data which is already reduced into coordinates. If the data has not yet been reduced and raw file contains observations, it is advisable to use the Raw Data Editor instead. This will allow the observed data to be recorded and reduced within the project.

On the *Point Code and Attributes* dialog box, codes assigned for each point will allow each imported point to be imported reading the Survey codes and code settings Library. This automatically sets point properties and layers for each point (Figure 5-21)t.

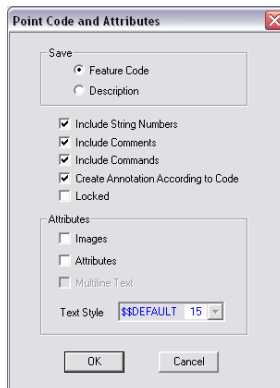


Figure 5-21. Set Point Codes and Attributes for Raw Files

Export

You can export data from a MAGNET Office project in several ways for use in other software and operations.

Save the data to be exported in a file format other than MAGNET Office (.ccx) format.

MAGNET Office can write to the file formats listed in Table 5-5

Table 5-5. File Formats (Write)

MAGNET Office Export File Formats	
civilcad 5 ASCII files	Genio files
AutoCAD files (*.dwg)	MCA files
AutoCAD files (*.dxf)	W3D files
ASCII Point files	Land XML files
MicroStation files (*.dgn)	TopSURV files (*.tsj)
3DMC Project files (*.tp3)	3DMC files (*.rd3, *.tn3, *.pt3, *.ln3)

To Export a File, do the following:

1. Click **File** ▶ **Export**.

The *Save As* dialog box appears.



To export Raw files, click **File Data Transfer**.

2. Select the appropriate file type from the **Save as type** drop-down menu.
3. Press **Save**.

The current project will be exported into the selected location.



Some import options may trigger the software to prompt for additional settings in order to be confirmed.

TopSURV On the *TopSURV* export dialog box, select either *All* or *Active* layers to be imported (Figure 5-22).



The TopSURV Utility must be installed to run this feature.



Figure 5-22. Export Options for TopSURV

Autodesk RealDWG Before export Autodesk RealDWG data, you must select how some of the data is to be treated. Generally, any data that displays and is active will be exported.

Make the following selection in the *Export AutoCAD DWG Options* dialog box (Figure 5-23 on page 5-21).

- Select From – select either All Layers or Active layers only to be imported.
- Point Heights – this selection may not be necessary if exported data is for drafting purposes only.
- Annotation – a property of an entity and will be converted to text when exported to AutoCAD. As additional text is created for these annotations, the user controls what layer the text will be exported into.
- Use Layer Translation Table – disables the Annotation option if this box is check marked.

- Export Triangles as: – exists in the project, the current DTM and contours will be exported if they display.
- Export Strings as: – export strings as either 2D Polylines or 3D Polylines. 3D polylines may not be necessary if exported data is for drafting purposes only.
- Distance Unit Conversion – select either No Conversion or To Millimeters, depending on whether the data is to be treated as ground or plan units.

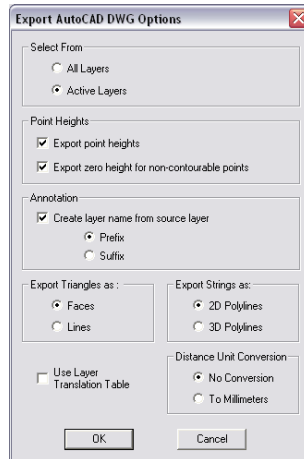


Figure 5-23. AutoCAD DWG Export Options

LandXML select the data to be exported from the *XML Export Selection* dialog box (Figure 5-24).



Figure 5-24. XML Export Options

GENIO On the *Moss Export Selection* dialog box, specify the data to be exported (Figure 5-25).

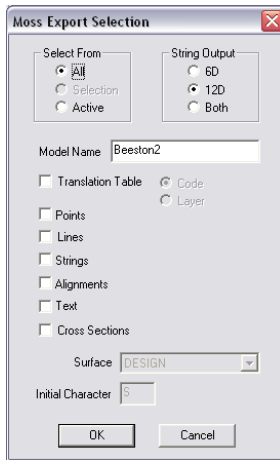


Figure 5-25. GENIO Export Options

ASCII For ASCII files you must define how points are exported on the *Export Points* dialog box in order to create a text file (Figure 5-26).

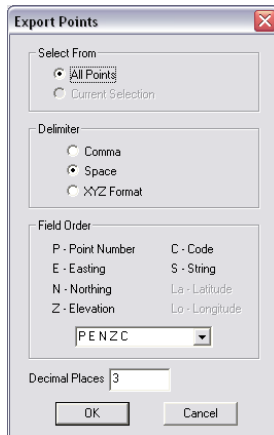


Figure 5-26. ASCII File Export Options

3DMC Project Select the data you want to export on the *3DMC Project* dialog box (Figure 5-27 on page 5-23).

The data can be viewed and selected by opening the tabs (Points, Lines, ALignment, DTM). If more than one export option is selected, multiple files will be generated for 3DMC (Figure 5-27).

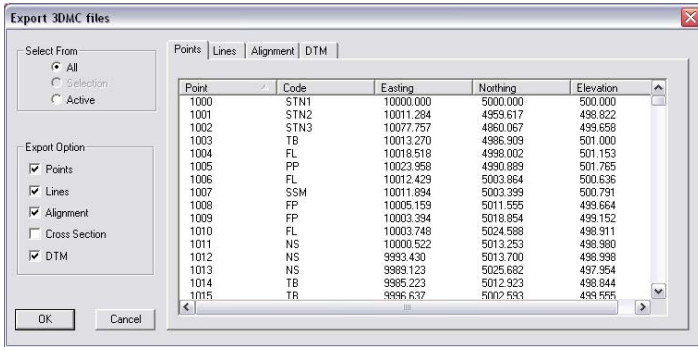


Figure 5-27. 3DMC Project Export Options

CivilSurv Specify the data needed to be exported on the *Export CivilSurv* dialog box. Road data can also be exported for this format for setout (Figure 5-28).

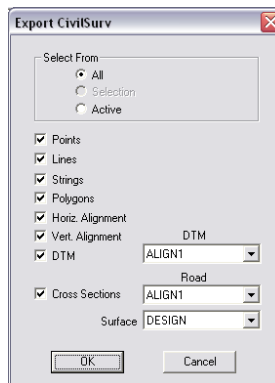


Figure 5-28. CivilSurv Export Options

RoadRunner On the *RoadRunner Export Selection* dialog box, (Figure 5-29 on page 5-24) Point, Road, and DTM data can be exported to Roadrunner. If all components are required, they must be exported individually.



Figure 5-29. RoadRunner Export Options

3D Viewer is included in the software, which allows you to view the DTM surface and data in 3D View. If an alignment exists in the project, you can virtually “drive” along the road (Figure 5-30).



Only one DTM surface can be exported at a time.



Figure 5-30. 3D Viewer Export Options

Export to Pocket-3D

Use the export to Pocket-3D command to export a 3DMC project file (*.tp3) to the mobile device. The transferred data is based upon use upon the user selections on the *Export 3DMC files* dialog box (Figure 5-31).

To export a 3DMC project file, click **File ► Export to Pocket 3D**. The *Export 3DMC files* dialog box displays (Figure 5-31).

1. Tap on either the Points, Lines, Alignment, or DTM tab to select the data you want to be included in the exported *.tp3 file.
2. In the *Select From* panel, choose one of the following:
 - All: enable to export all of the design data available for transfer within the project. Visibility of this data is not a prerequisite. Design data contained in layers that are turned off will still be available for transfer using this option.
 - Selection: enable to only select supported data for transfer within a selection set for inclusion within the exported *.tp3 file.
 - Active: enable to only select supported visible data within the main survey view for inclusion within the exported *.tp3 file.

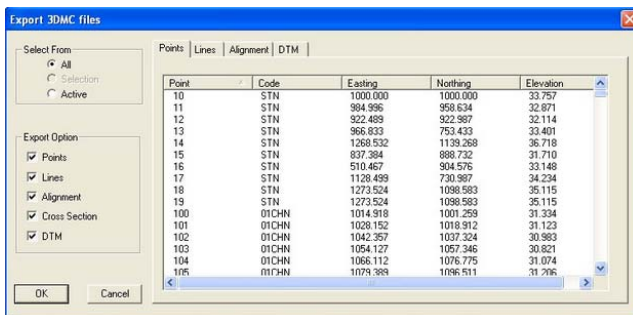


Figure 5-31. Export a 3DMC Project File

Genio Translation Table

Use the Genio Translation Table command to define settings for the export of data to a Moss Genio file. This produces a file that complies with user-defined requirements. Set the Genio table preferences before using the **Export** command to save the Genio file.

To use the Genio table preferences, do the following:

1. Open the project from which data will be exported.
2. Set the active layers, contours, and annotations as required in the exported file.
3. Click **File ▶ Genio Translation Table**.
The *Moss Export Lookup Table* dialog box appears. (Figure 5-32)



Figure 5-32. Translation Table for Export to Genio Files

This table converts MAGNET Office layers and codes to specified Moss Strings.

4. Set up the **Moss Export Lookup Table** for the required formats using Table 5-6.

Table 5-6. Genio Translation Requirements

Field Name	Field Description
Entity	Select either Alignment, Line, Point, or String from the drop-down list.
Code/Name	Mainly used if you have already created MAGNET Office strings or alignments and named them. These MAGNET Office strings will be converted to the Moss String specified.

Table 5-6. Genio Translation Requirements (Continued)

Field Name	Field Description
Layer	Allows you to select a layer from within the MAGNET Office project to be converted into the specified Moss string name.
Moss String	Enter a name for the Moss String.
Save To Lib	Copies the current table to the library under a specified name.
Copy From Lib	Updates the table with one from the library. A message appears to ask if you want to retain the existing Acad Layer names. Select a library table to copy into the table.
Insert	Press to insert an additional row before the currently selected row in the table.
Delete	Press to delete the currently selected row from the table.
Restore	Press to restore the table to the settings before items were edited.

5. When data entry is complete, press **OK** to save the new table parameters or press **Cancel** to quit.
6. After setting up the table, export the selected data to a Moss string or Genio (.gen) file by clicking **File ▶ Export**.
The *Save As* dialog box appears.
7. Locate and select the Genio file, and then press **Save**.
The *Moss Export Selection* dialog box displays (Figure 5-33 on page 5-28).

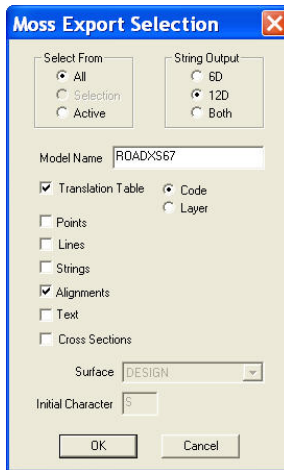


Figure 5-33. Moss Export Selection

Table 5-7. Moss Export Selection Dialog Box Descriptions

Field Name	Field Description
Select From	<ul style="list-style-type: none"> • All – enable to export all of the design data available for transfer within the project. Visibility of this data is not a prerequisite. Design data contained in layers that are turned off will still be available for selection to be transferred using this option. • Selection – (when available) enable to only select supported data for transfer within a selection set for inclusion within the exported *.tp3 file. • Active – enable to only select supported visible data within the main survey view for inclusion within the exported *.tp3 file.
String Output	Select either 6D, 12D, or Both.
Model Name	The name of the file being exported.
Translation Table	Check mark this box to use the Moss string naming conventions selected from the adjacent field; select Code or Layer
Points	Check mark this box to export points using the name of the point.
Line	Check mark this box to export lines using the name of the line.

Table 5-7. Moss Export Selection Dialog Box Descriptions (Continued)

Field Name	Field Description
Strings	Check mark this box to export strings using the name of the string.
Alignments	Check mark this box to export alignments using the name of the alignment.
Text	Check mark this box to export text.
Cross sections	Check mark this box to export cross sections. Select a Surface type and enter an Initial Character.

Autocad Layer Translation Table

Use the Autocad Layer Translation Table command to specify the Autocad layer to be used for translation in place of the MAGNET Office layers used in the project. When opening the Layer Translation Table for the first time in the current project, it is automatically filled with the active layers in that project. If the Autocad layer name used in the table does not exist in the definition table, then the corresponding MAGNET Office layer name will be marked with a **0** setting.



The Autocad default layer 0 cannot be deleted or renamed. All exported files to *.dxf* and *.dwg* must have a 0 layer.

To use the Autocad Layer Translation Table command, do the following:

1. Click **File ▶ Autocad Layer Translation Table**.

The *Layer Translation Table* dialog box appears (Figure 5-34 on page 5-30).

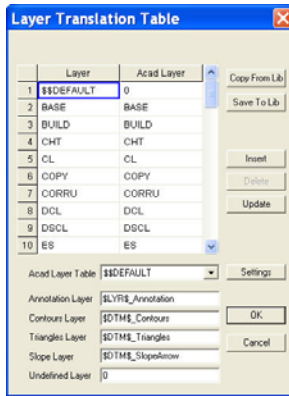


Figure 5-34. Layer Translation Table

The first column, *Layer*, lists the layers in the MAGNET Office project. The second column, *Acad Layer*, lists the corresponding layer to which the MAGNET Office layer will be translated.

2. Modify the table as necessary. For field descriptions, see Table 5-8. For button descriptions, see Table 5-9.
3. Press **OK** to accept your modifications and close the table.

Table 5-8. Layer Translation Table Field Descriptions

Field Name	Field Description
Acad Layer Table	List of definition tables from the library.
Annotation Layer	All annotations are placed into the nominated layer name. As a default, they are placed in \$LYR\$_Annotation, where \$LYR\$ is the layer name. All short line/arc tables are placed into the default layer 0.
Contours Layer Triangles Layer Slope Layer	All contour lines, triangles, and slope arrows are placed into the nominated layer. By default, they are placed in \$DTM\$_Contours/Triangles/SlopeArrow, where \$DTM\$ is the DTM name from which the contours, triangles, and slope arrows are determined.
Undefined Layer	Defaults to layer 0. All entities in non-specified Autocad layers in the table are put into this layer.

Table 5-9. Layer Translation Table Button Descriptions

Button Name	Button Description
Copy From Lib	Updates the table with one from the library. A message appears asking if you want to retain the existing Acad Layer names. Select a library table to copy into the table.
Save To Lib	Copies the current table to the library under a specified name.
Insert	Creates a new row above the selected row.
Delete	Deletes all selected rows from the table.
Update	Updates table to include all layers in the current project.
Settings	Opens the Layer Definition Table.
OK	Accepts and saves all new changes to the table.
Cancel	Closes the table without saving changes.

Synchronization Utility

The Synchronization utility will keep TopSURV or Pocket-3D synchronized with MAGNET Office. After connecting your field controller to the PC using ActiveSynch, the synchronization utility will automatically launch. To use the synchronization utility, do the following:

1. Click **File ► Synchronization Utility** on the File menu. The *Desktop-CE Device Synchronization* dialog box displays (Figure 5-35).

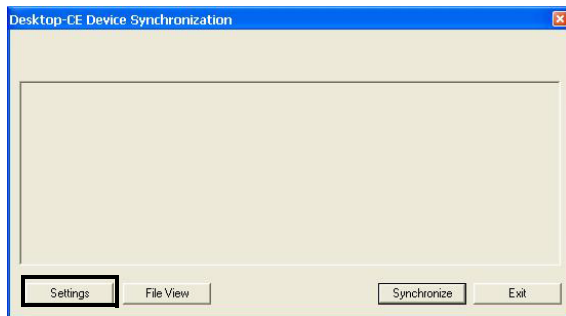


Figure 5-35. Desktop Device Synchronization Utility Menu

Synchronization Settings

1. Press **Settings** on the *Desktop-CE Device Synchronization* dialog box (Figure 5-35 on page 5-31). The *Options* dialog box displays (Figure 5-36).

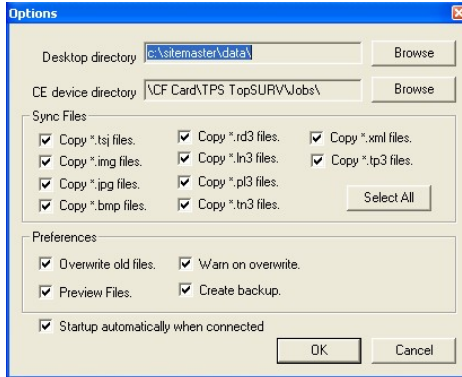


Figure 5-36. Options Dialog Box

2. On the *Options* dialog box, do the following to specify the location of the files to be synchronized, as well as the file type.
 - Desktop directory – press the **Browse** button to specify to specify the directory for synchronization on your PC device (Figure 5-41 on page 5-38).
 - CE device directory – press the **Browse** button to specify the directory for synchronization on your mobile device (Figure 5-35).
 - Sync Files – on the *Synch Files* panel, place a check mark in the box opposite the file type you want to be synchronized; select either *Copy *.tsj files*, *Copy *.img files*, *Copy *.jpa files*, *Copy *.bmp files*, *Copy *.rd3 files*, *Copy *.ln3 files*, *Copy *.p3 files*, *Copy *.tns files*, *Copy *.xml files*, or *Copy *.tp3 files*; press **Select All** to synchronize all file types.

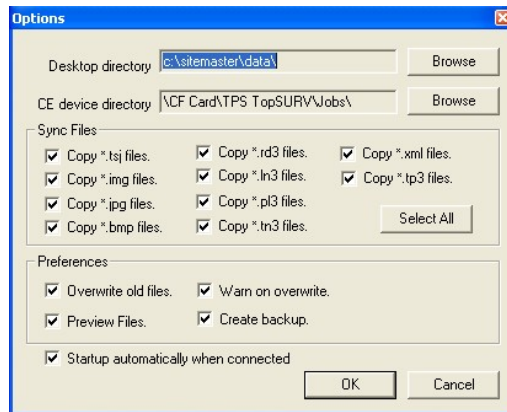


Figure 5-37. Options Synchronization Utility Screen

Files will be transferred both from the mobile device to the desktop computer and vice versa. File types with no check mark of the same name will not be synchronized.



You can also press **Settings** on the *File Transfer - Copy From* dialog box (Figure 5-39 on page 5-35) to display the *Options* dialog box.

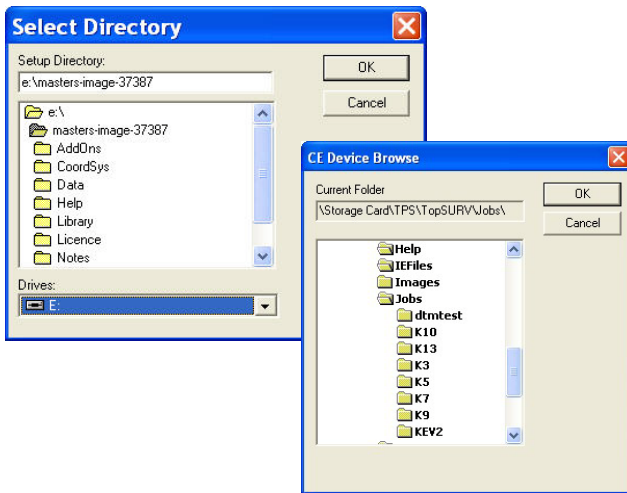


Figure 5-38. Desktop and CE Device Directory

- Preferences – starts up automatically when connected. When toggled on, if a mobile device connection is detected, the synchronization utility will start up and display the *file transfer – Copy From* dialog box (Figure 5-39 on page 5-35):
 - Overwrite old files: check mark this box to replace older files with a newer file of the same name and type during synchronization.
 - Warn on overwrite: check mark this box to give a warning before you overwrite a file.
 - Create backup: check mark this box to create a backup of the file you have made.
 - Preview Files: check mark this box to preview files before saving.
- **OK** – saves and returns to the *Options* dialog box (Figure 5-39 on page 5-35).
- **Cancel** – cancels the synchronization process.

Desktop-CE Device Synchronization

1. Press the **File View** button on the *Desktop-CE Device Synchronization* dialog box (Figure 5-35 on page 5-31). The *File Transfer - Copy From* dialog box appears (Figure 5-39).



If the synchronization utility has been closed, click **File ▶ Synchronization Utility** to launch it again.

2. On the *File Transfer – Copy From* dialog box (Figure 5-39), place a check mark next to the file(s) you want to transfer. The same procedure can be followed for both transferring data to a computer, downloading files, or uploading files to a field controller. Transferring specific files can be used in lieu of synchronization in cases where synchronization is not appropriate.
3. To transfer all files, check mark the *Select all file(s)* box in the upper right-hand corner (Figure 5-39).

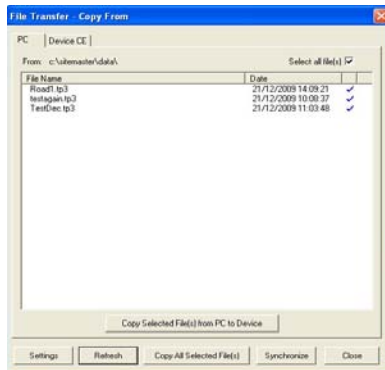


Figure 5-39. File Transfer – Copy From Dialog Box

Synchronization Setting

Press Synchronization to copy new files back and forth from the mobile device to the PC, which keeps the mobile device and the PC synchronized, based on files.

Data Transfer

You can transfer a set of *XYZ* coordinates or road alignment details from your computer to a data recorder. A range of data recorders is supported; however, you must ensure the correct communication settings are applied. The settings on the data recorder must match the settings on the computer.

To transfer data, do the following:

1. Open the project from which data will be transferred.
2. Click **File ▶ Data Transfer**.

The *Transfer Data* dialog box appears. (Figure 5-40)

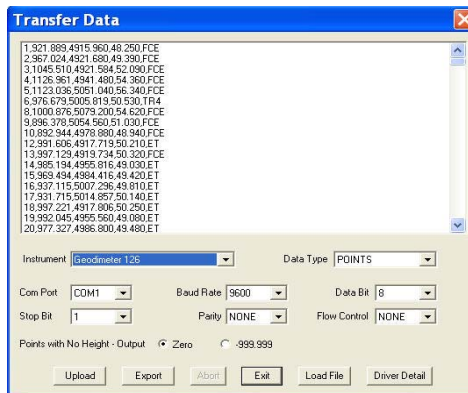


Figure 5-40. Transfer Data – Points Ready for Uploading

3. Set the parameters on the *Transfer Data* dialog box (Figure 5-40) to match the communication parameters defined on the data recorder. You may need to refer to the user manual supplied by the data recorder manufacturer. MAGNET Office saves these settings for future upload sessions. For field descriptions, see Table 5-10 on page 5-37.
4. When the data is set, do one of the following:
 - Press **Upload** to begin transferring data.
 - Press **Export** to create a file of the selected data or to export the file to a mobile device.
 - Press **Abort** to terminate the data transfer.

- Press **Load File** to upload data from a previously saved ASCII Points (.pts) file or **Upload** (.upl) file.
 - Press **Driver Detail** to display the current instrument driver.
5. Make sure the data recorder is ready to receive information, and press **Receive** or a similar command on the instrument.
 6. To finish the data transfer, press **Exit**.

Table 5-10. Transfer Data Descriptions

Field Name	Field Description
Instrument	The instrument manufacturer and data format.
Data Type	The type of data to transfer. For most instruments, this should be set to Points as other types may not be supported.
Com Port	The serial port on the computer that is connected to the data recorder.
Baud Rate	Determines how fast data is transferred between instruments that use serial communication. This rate must be the same on the computer and the instrument.
Data Bit	Determines the order of transmission from the least to the most significant bit. This rate must be the same on the computer and the instrument.
Stop Bit	The last part of a character frame consists of 1, 1.5, or 2 stop bits. This rate must be the same on the computer and the instrument.
Parity	Detects errors and ensures data is transferred accurately. This rate must be the same on the computer and the instrument.
Flow Control	Tells one device to tell another device to stop sending data for a while. This rate must be the same on the computer and the instrument.
Points with No Height – Output	The value entered here will be used on elevation property of the point. For example, you may want to set points with no heights to -9999999.

Print Setup

MAGNET Office prints projects according to the settings you have defined in the standard *Microsoft Windows Print Setup* dialog box. The printed image is the data displayed on the screen and is fitted to

the selected page size. The printout is bordered with a box and is captioned with various details, including the date and scale of the printout.

To define printer or plotter settings, do the following:

1. Click **File ▶ Print Setup** (Figure 5-41).

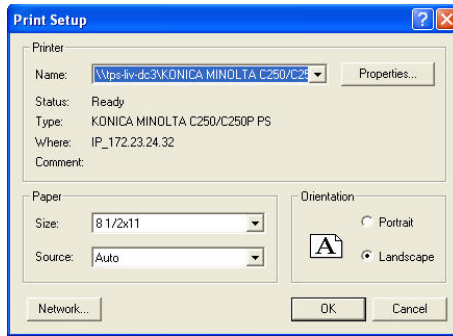


Figure 5-41. Print Setup

2. Make sure you have the correct settings defined for the printer or plotter, including the printer name, paper size, and paper orientation.
3. Click **Properties** to access the printer driver details and modify the printer or plotter settings if necessary.
4. When you have finished, press **OK**.

Print

Map the data to fit the extents of the drawing area of the Survey View window. This defines the extents of the data for the printout.

To print a project:

1. Click **File ▶ Print** or press the shortcut keys **Ctrl+P**.
The **Print** dialog box appears (Figure 5-42 on page 5-39).

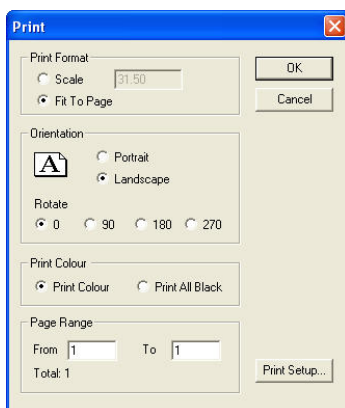



Figure 5-42. Print Dialog Box

2. Make sure you have the correct settings defined for the printer or plotter, and then press **OK**.

Printing Projects Saved in the Drawings View When a linked or unlinked drawing is saved as a project in the Drawings View, the 2D drawing is written to a separate project and displayed in the Survey View, where it can be edited.

The drawing data is displayed, enclosed within the title block originally defined for the Drawing. You can print this drawing to scale, and the printout will contain the data and the title block.

Exit

To close the MAGNET Office program, make sure you have saved your project(s), and then click **File** ► **Exit** or the **Close**  icon at the top right corner of the dialog box.

Survey View: Edit Menu

This section describes the different Survey View menu commands on the **Edit** menu located on the main menu bar (Figure 6-1).

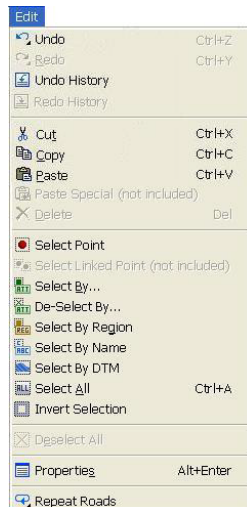


Figure 6-1. Survey View – Edit Menu

In This Section:

- “Undo” on page 6-2
- “Redo” on page 6-2
- “Undo History” on page 6-3
- “Redo History” on page 6-3
- “Cut” on page 6-4
- “Copy” on page 6-4
- “Paste” on page 6-6
- “Delete” on page 6-8
- “Select Point” on page 6-9

- “Select By” on page 6-10
- “De-Select By” on page 6-18
- “Select By” on page 6-10
- “Select By Name” on page 6-20
- “Select By DTM” on page 6-21
- “Select All” on page 6-22
- “Invert Selection” on page 6-22
- “DeSelect All” on page 6-23
- “Edit Properties” on page 6-23
- “Repeat” on page 6-24

Undo

Use the Undo command to eliminate the last action performed. The creation of entities is recorded in sequence during a working session, and this sequence can be reversed to “undo” those actions. This allows you to easily correct errors. Entities with links, such as circle center points, will be removed in the same manner as they were created.

The number of steps recorded during a routine is unlimited, and the **Undo** command can step back to the beginning of the routine.

To undo an action, click **Edit ▶ Undo**, or press **Ctrl+Z**.

You can also right-click the mouse, and select **Undo** from the shortcut menu.

Redo

You must have performed an action and used the **Undo** command before the **Redo** command becomes available. Redo moves forward through the commands that were reversed by the **Undo** command.

To redo an action, click **Edit ▶ Redo** or press **Ctrl+Y**.

Undo History

The Undo command displays a list of executed commands. To undo a command, select it from the list (all previously executed commands are also highlighted), and click **OK**. The highlighted commands are removed from the display area (Figure 6-2).

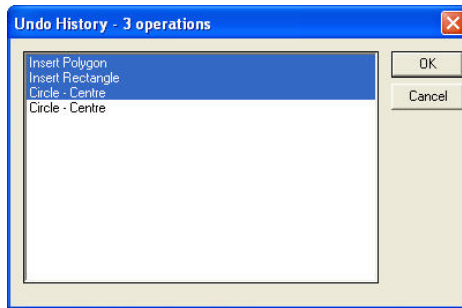


Figure 6-2. Undo History

Redo History

The Redo History command displays a list of commands that were “undone.” To redo these commands, select one from the list (all previously undone commands are also highlighted), and click **OK**. The highlighted commands reappear in the display area (Figure 6-3 on page 6-3).

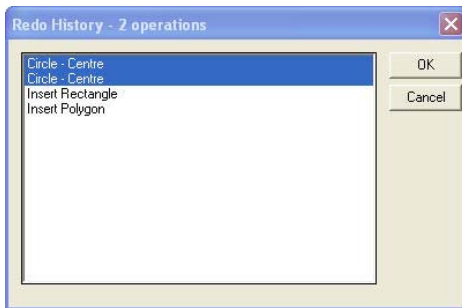



Figure 6-3. Undo History

Cut

Use the Cut command to remove selected entities from the data set similar to the way the **Delete** command and places the cut data onto the clipboard.

To remove an entity from a data set, do the following:

1. Select the data or highlight it by dragging a window around it with the mouse.
 - Do one of the following:
 - Click **Edit ▶ Cut**
 - Press **Ctrl+X**
 - Click the **Cut**  icon
2. To restore the cut data, click **Edit ▶ Undo**.
3. To paste the cut data from the clipboard to another position or into another drawing, click **Edit ▶ Paste**. For more information, see “Paste” on page 6-6.

Copy



Use the Copy command to copy selected entities to the clipboard. The entities must be selected before this command becomes available. The selected data may be copied from one project and pasted into another at the same coordinates.

To copy a single entity to another position within the same MAGNET Office project:


1. Select the entity by clicking on it or dragging a window around it.
2. Press and hold the **Ctrl** key.
3. Click on the entity’s center (yellow) handle, and drag it to a new position.
4. Release the mouse to save the copied entity.

To copy a single entity to another project within MAGNET Office:

1. Select the entity by clicking on it or dragging a window around it.

2. Click **Edit ▶ Copy** or click the **Copy**  icon on the **Standard** toolbar.
3. In the other project in which you want to place the copied entity, click **Edit ▶ Paste** or click the **Paste**  icon on the **Standard** toolbar.
4. If you are pasting into another view, the **Duplicate Point Name Detected** window appears. Select the **Renumber all points** check box (recommended) or choose to duplicate the point name, and click **OK**.
5. If you do not see the entity you pasted, then zoom out by using the options on the right-click shortcut menu or the icons on the **Standard** toolbar.
6. To move an entity within the same project:
 1. Select the entity by clicking on it or dragging a window around it.
 2. Click **Modify ▶ Move**.

The entity becomes highlighted with a yellow box around it.

3. Click on the center point or handle.
The pointer changes to the **Move Entity**  pointer.
4. Drag the entity to a new position, or enter the new coordinates into the input bar and then press **Enter**.
The **Move Confirmation** window appears (Figure 6-4).

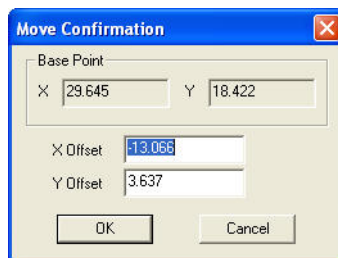


Figure 6-4. Move Confirmation Window

5. Click **OK** to place the entity into the new position.

You can copy text from one project and paste it into another project at the same coordinates. If the second project already contains data, the points will be renumbered to prevent overwriting existing points.

To copy and paste text from other compatible applications:

1. Copy the text from the other application.
2. Set the text style.
3. Click **Insert ▶ Text ▶ Text**.
4. Click **Edit ▶ Paste**.

If the text is pasted into MAGNET Office in one long text string (i.e., word wrap active in other applications) then double-click the inserted text and break the text string into multiple lines using the **Enter** key at suitable places in the text string.

Paste

Use the Paste command to paste data from the clipboard into the current project in the Survey View.

When transferring data from one position to another, select the entities and use the **Cut** or **Copy** command. The data is placed into an area of memory called the clipboard. When the **Paste** command is used, these entities are read from the clipboard and placed into the currently active project window at the same coordinate position. You can move the data to another coordinate position. The attributes and properties (coordinates, dimensions, color, line style, etc.) will remain the same.

The **Paste** command is only available if data was cut or copied to the clipboard.

Data may be cut or copied from one project and pasted to another. The data will be pasted into the second project at the same coordinates. If the second project already contains data, the points will be renumbered to prevent overwriting existing points.




Instead of cutting data to paste to another position, use the **Move** option. Right-click in the window, and select **Move Selection** from the shortcut menu.

To paste data, do the following:

1. Select the entity you want to paste.

2. Do one of the following to paste the selected entity:

- Click **Edit ▸ Paste**.
- Press **Ctrl+V**.
- Click the **Paste**  icon on the **Standard** toolbar.

All selected entities are pasted from the clipboard into the current project at the same position as the cut or copied data.

If duplicate point names (numbers) are found, then the **Duplicate Point Name Detected** dialog box appears (Figure 6-5 on page 6-7).

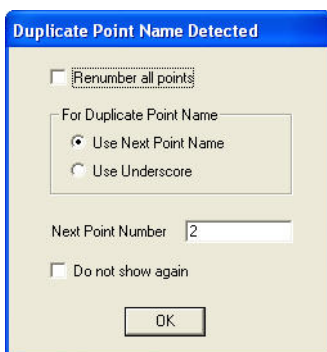


Figure 6-5. Duplicate Point Name Detected

3. Check mark the **Renumber all points** check box to renumber all of the points in the pasted data beginning with the point number defined in the **Next Point Number** field.

If the **Renumber all points** check box is not selected, then the existing point numbers will be used unless the number already exists in the project. If the point number does exist, then the pasted point will be allocated another number beginning with the point number defined in the **Next Point Number** field.

4. Select the **Use Next Point Name** option to renumber any duplicated points beginning with the point number defined in the **Next Point Number** field.
5. Select the **Use Underscore** option to renumber any duplicated points with the original point number followed by an underscore

and one or more numbers to define the number of copies for this point number.

The **Next Point Number** is used for any points that need new numbers. This is read from the **Default Attributes** page on the **General Settings** (Hot Button) dialog box.

6. Check mark the **Do not show again** box to accept the defined settings for additional data pasting. The next point number will be taken automatically from the next point number in the project.
7. Click **OK**. The data on the clipboard is displayed in a yellow selection box over the copied data or in the same position as the removed (cut) data.
8. To modify the position of the pasted data, press and hold the **Shift** key, select the yellow handle on the lower left of the window, and drag the handle and data to a new position.



If the **Paste** command is abandoned after the data has been displayed on the screen, click the **Undo** command to remove the pasted data and prevent duplication of the pasted data over the existing data.

Pasting Text To paste text from a compatible application, do the following:


After text has been selected and the copy option has been chosen, in order to place the text into Survey view at a user-defined point, you must click the **Insert ▶ Text ▶ Text** option, select the location of the text, then click **Edit ▶ Paste**.

NOTE: If you choose text from another program, then come into MAGNET Office, you cannot paste in text until you have chosen the **Insert ▶ Text ▶ Text** option.

Delete

The Delete command deletes an entity or entities from the current project database.

To delete entities, do the following:

1. Select the entity you want to delete.
2. Click **Edit ▶ Delete**, or click the **Erase**  icon on the **Standard** toolbar.
3. To restore the deleted data, click **Edit ▶ Undo**.

Select Point

Use the Select Point command to select and edit individual points. You can select a point by clicking it or by entering a known point number.

Do either of the following to enter known numbers:

1. Click **Edit ▶ Select Point**.


The input bar displays at the lower left-side of the window (Figure 6-6).



The input bar consists of three main sections: 'Number' with the value '1', 'East' with the value '518.212', and 'North' with the value '100.931'. Each section has a small square icon to its right.

Figure 6-6. Input Bar

2. Enter the required point number in the **Number** field, and press **Enter** to highlight the point with a yellow selection box.

- Click the **Enquire (Select) Point**  icon.

The input bar displays at the lower left-side of the window (Figure 6-7).



The input bar is identical to Figure 6-6, showing 'Number' (1), 'East' (518.212), and 'North' (100.931) fields.

Figure 6-7. Input Bar

3. Select a point or enter a point number in the **Point Name** field.
4. Press **Enter** or left-click to highlight the point and display the **Edit Point** properties dialog box.



You cannot edit properties displayed in fields that are gray.

Select By

Use the **Select By** command to apply property filters to selected entities. Selections are highlighted in yellow and enclosed in a dashed yellow box.

When you select an entity and click **Edit ▶ Select By**, the *Select by* dialog box appears (Figure 6-8).

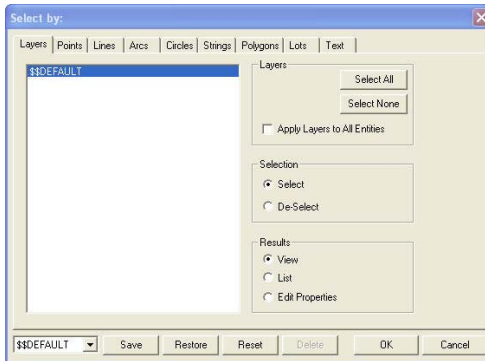


Figure 6-8. Select By Dialog Box

This dialog box has nine tabs: a layer tab and one tab for each type of entity. The required property filters are defined on one or more tabs, and the required layers are highlighted. The selected entity is highlighted in yellow in a selection box on the screen and is ready for further action.

Layers Tab

On the **Layers** tab, only the active layers are listed. Select the layer(s) to which you want to apply a filter (Figure 6-9 on page 6-11).

To select a layer, do one of the following:

- A single layer – point and click.
- Consecutive layers – click the first layer, press and hold the **Shift** key, and then click the last consecutive layer.
- Dispersed layers – click the first layer, and then press and hold the **Ctrl** key as you click each individual layer.

- All layers – click **Select All**.
- No layers – click **Select None**.

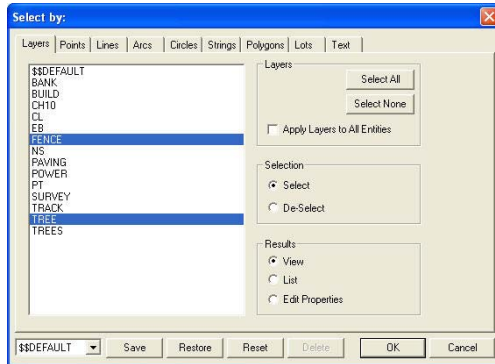


Figure 6-9. Select By Dialog Box – Layers Tab

To reset the layers to how they appeared when you first opened the **Select by** dialog box, click **Reset**. To view the layers you last selected, click **Restore**.

When you are finished, you can:

- Click **Save** to save your selection(s).
- Click **OK** to return to the Survey View.
- Click another tab to set more filters.

Points Tab

To select points on selected layers, press the *Points* tab (Figure 6-10 on page 6-12) and check mark the **Select Points** box. Apply filters for the points by name, code, height, mark, color, or any combination. Points matching these property filters will be selected. For a description of the fields, see Table 6-1 on page 6-12.

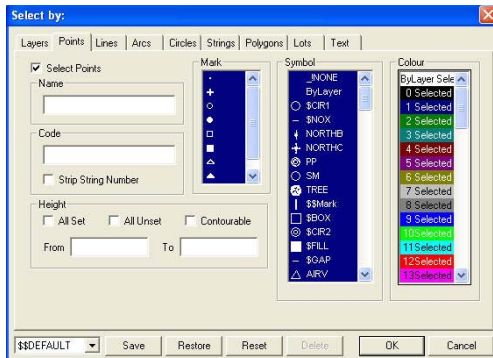


Figure 6-10. Select By Dialog Box – Points Tab

Table 6-1. Points Tab Field Descriptions

Field Name	Field Description
Name	To select points by name (alphanumeric ID or point number), type a name separated by commas or a name range separated by a hyphen. For example: 50-54, 95 will select points 50, 51, 52, 53, 54, and 95.
Code	To select points by code, enter a code, individual codes separated by commas, or an alphabetical code range separated by a hyphen. This range can be just the start characters of the code range. For example: R-T will elect points with any code starting with R or S but none of those starting with T unless a code of the single character T is available.
Strip String Number	Select this check box to remove the string number from the code to aid selection.
Height All Set	Choose this check box to select all points with the elevation flag on (set).
All Unset	Choose this check box to select all points with the elevation flag off (unset).
Contourable	Choose this check box to select all points that are flagged for use when creating a surface.
From and To	To select points by height, enter the lowest elevation in the range in the From field followed by the highest elevation in the range in the To field.
Mark	To select points by point mark, select individual marks or a range of marks.

Table 6-1. Points Tab Field Descriptions (Continued)

Field Name	Field Description
Symbol	To select points by point symbols, select individual symbols or a range of symbols.
Color	To select points by color, select individual colors or a range of colors.

To reset the layers to how they appeared when you first opened the **Select by** dialog box, click **Reset**. To view the layers you last selected, click **Restore**.

When you are finished, you can:

- Press **Save** to save your selection(s).
- Press **OK** to return to the Survey View.
- Click on another tab to set more filters.

Lines, Arcs, Circles, and Strings

The filters for Lines, Arcs, Circles, and Strings are similar. Click on the *Line* tab (Figure 6-11 on page 6-14), and check mark the Select Lines, Arcs, Circles, or Strings box, depending on which tab is opened. Apply filters by line style, line thickness, or color. Lines, arcs, circles, and strings matching these property filters will be selected. For a description of the fields, see Table 6-2.

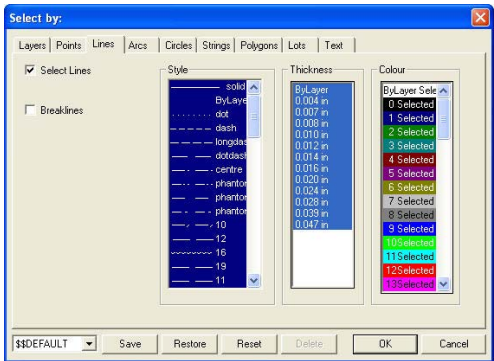


Figure 6-11. Select By Dialog Box – Line Tab

Table 6-2. Tab Field Descriptions

Field Name	Field Description
Breaklines	To select only the breaklines of a particular entity. For example: If you choose the Select Lines check box and the Breaklines check box, then only the lines that have been designated as breaklines will be selected.
Line Style	To select an entity by line style, select one or more line styles
Line Thickness	To select an entity by line thickness, select one or more thickness options.
Color	To select an entity by color, select one or more colors.
Code (Strings Tab only)	To select strings by code name, type a string code, individual codes separated by commas, or an alphabetical code range separated by a hyphen.

To reset the layers to how they appeared when you first opened the *Select by* dialog box, press **Reset**. To view the layers you last selected, press **Restore**.

When you are finished, you can:

- Press **Save** to save your selection(s).
- Press **OK** to return to the Survey View.
- Click on another tab to set more filters.

Polygons Tab

To select polygons on the chosen layers, click on the *Polygons* tab and check mark the **Select Polygons** box (Figure 6-12). Apply filters for the polygon by line style, line thickness, color, boundary, or any combination. Polygons matching these property filters will be selected. For more information, see Table 6-3.

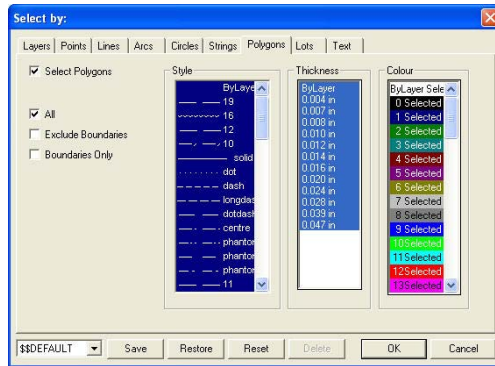


Figure 6-12. Select By Dialog Box – Polygons Tab

Table 6-3. Polygons Tab Field Descriptions

Field Name	Field Description
Line Style	To select an entity by line style, select one or more line styles.
Line Thickness	To select an entity by line thickness, select one or more thickness option.
Color	To select an entity by color, select one or more colors.
All	To select all polygons and boundaries, select this check box.
Exclude Boundaries	To exclude boundaries from the polygon selection, select this check box.
Boundaries Only	To select only boundaries in the polygon selection, select this check box.

To reset the layers to how they appeared when you first opened the **Select by** dialog box, click **Reset**. To view the layers you last selected, click **Restore**.

When you are finished, you can:

- Press **Save** to save your selection(s).
- Press **OK** to return to the Survey View.
- Click on another tab to set more filters.

Lots Tab

To select lots on the chosen layers, click on the **Lots** tab and check mark the **Select Lots** box (Figure 6-13). Apply filters for the lots by line style, line thickness, color, code, DP Number, or any combination. Lots matching these property filters will be selected. For more information, see Table 6-4.

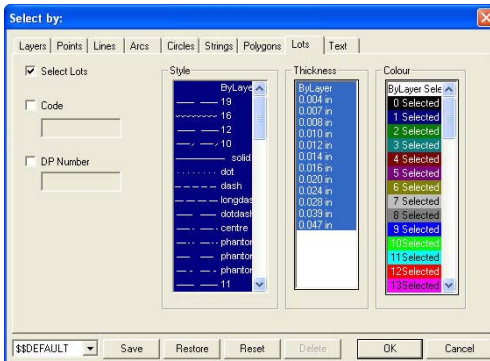


Figure 6-13. Select By Dialog Box – Lots Tab

Table 6-4. Lots Tab Field Descriptions

Field	Description
Line Style	To select an entity by line style, select one or more line styles
Line Thickness	To select an entity by line thickness, select one or more thickness options.
Color	To select an entity by color, select one or more colors.
Code	To select lots by code, type the required code or code range.
DP Number	To select lots by DP number, type the DP number or number range.

To reset the layers to how they appeared when you first opened the **Select by** dialog box, click **Reset**. To view the layers you last selected, click **Restore**.

When you are finished, you can:

- Press **Save** to save your selection(s).
- Press **OK** to return to the Survey View.
- Click on another tab to set more filters.

Text Tab

To select text on the chosen layers, click on the **Text** tab and check mark the **Select Text** box (Figure 6-14). Apply filters for the text by font, character size, color, style, or any combination. Text matching these property filters will be selected. For a description of the fields, see Table 6-5.

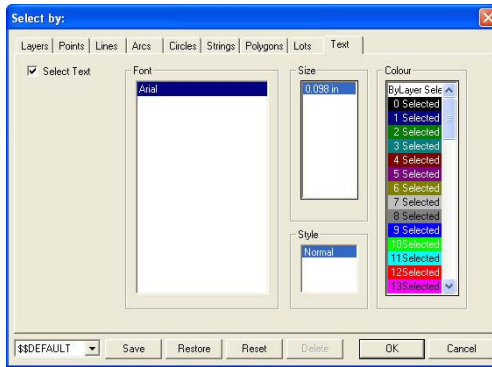


Figure 6-14. Select By Dialog Box – Text Tab

Table 6-5. Text Tab Field Descriptions

Field	Description
Font	To select the text by font, select one or more font options.
Size	To select the text by size, select one or more size options.
Color	To select the text by color, select one or more colors.
Style	To select the text by style, select one or more styles.
DP Number	To select lots by DP number, type the DP number or number range.

To reset the layers to how they appeared when you first opened the **Select by** dialog box, press **Reset**. To view the layers you last selected, press **Restore**.

When you are finished, you can:

- Press **Save** to save your selection(s).
- Press **OK** to return to the Survey View.
- Click on another tab to set more filters.

De-Select By

To use the De-Select By command, click **Edit ▶ De-Select By**. This command works in the same way the **Select By** command works, however, **De-Select By** removes an entity from a selection by entity type, such as breaklines.

The *De-Select By* dialog box appears (Figure 6-15).

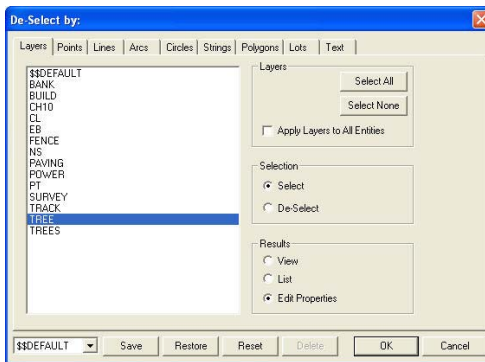


Figure 6-15. De-Select By Dialog Box

Select an entity, and then click **OK**. For a description of the dialog box, see “Select By” on page 6-10.

Select By Region

Use the **Select by Region** command to select entities using one of three methods:

- By using the mouse to draw a string around the data to define an enclosure or fence.
- By dragging a window around the data, and then selecting only one or two entity types in that window.
- By using an existing boundary as the enclosure.

To use an enclosure or fence to select entities, do the following:

1. Click **Edit ▶ Select By Region**.

The *Select By Region* dialog box appears (Figure 6-16).

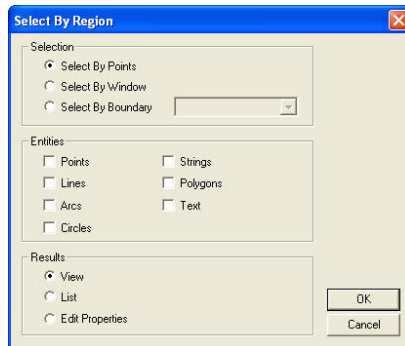


Figure 6-16. Select By Region Dialog Box

2. Under the *Selection* panel, choose one of the following options:
 - **Select By Points** – Choose this option to select a region by defining points on a string. This enables you to draw an enclosure, fence, or lasso around entities.
 - **Select By Window** – Choose this option to select a region by window. This enables you to drag a window around entities.
 - **Select By Boundary** – Choose this option to select data within a boundary. This enables you to select the existing boundary from a list.
3. Under the *Entities* panel, select the required entities. The entities that were not selected will be filtered out.

4. Under the **Results** panel, select how you would like the final results output or if you want to edit the results properties.
5. Click **OK**.

Select By Name

Use the Select By Name command to select entities that require or provide the option of naming. These entities are alignments, strings, boundaries, lots, and pads.

To use the Select By Name command, do the following:

1. Click **Edit ▶ Select By Name**.

The *Select By Name* dialog box appears (Figure 6-17).

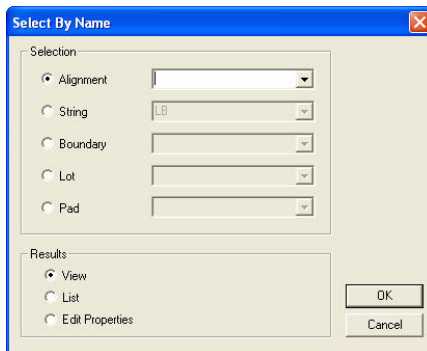


Figure 6-17. Select By Name

2. In the **Selection** panel, choose an entity and then select a name from the drop-down entry box.
3. In the **Results** panel, select one of the following:
 - **View** – select to highlights the named entity you selected on the screen.
 - **List** – select to list the important details of the named entity in a report.
 - **Edit Properties** – select to opens the edit dialog box for modifications.
4. Click **OK** to save and exit to the main screen.

Select By DTM

Use the Select By DTM command to select points that are within a specified depth difference range from that of a reference DTM. This difference can be entirely above or below a surface or a depth range that spans a distance below and above a surface.

1. Click **Edit** ► **Select By DTM** from the **Edit** menu. The *Select By DTM* dialog box appears (Figure 6-18).

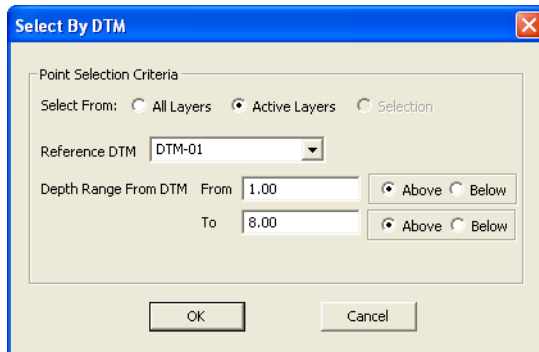


Figure 6-18. Select by DTM Dialog Box

2. Select the following options from the *Select by DTM* dialog box (Figure 6-19 on page 6-22):
 - **Select From** – select either *All Layers* or *Active Layers*.
Selection not available.
All Layers selects points within the specified depth range from all layers.
Active Layers selects point within the specified depth range from active layers only.
Selection selects points within a predetermined selection set. The selection must be created before running the command.
 - **Reference DTM** – specifies the reference DTM. All distances specified will be above or below this DTM.
 - **Depth Range From DTM** – Defines the depth range for point selection
 - **From/To:** enter a minimum depth (From) or a maximum depth (To) above or below the reference DTM.

- **OK** – press to select the points within the defined depth zone to display on screen.

Use the right-click context menu **Properties** option or click **Modify ▶ Multiple Point Edit** to edit point properties.



While in any option, you can check out the context menu for additional functions. To access the context menu, click the right mouse button.

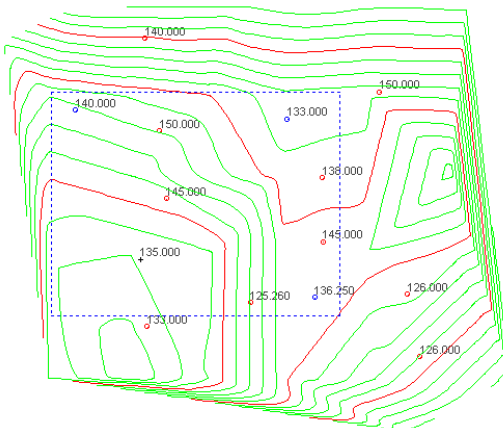


Figure 6-19. Example – Select Points by DTM

Select All

To select all of the entities currently displayed on the active layers, click **Edit ▶ Select All** or press **Ctrl+A**.

If you right-click on the selection, a menu of shortcut options appears, depending on the selected entity.

Invert Selection

Use the Invert Selection command to clear the selected entities and select all of the other entities on the active layer. This command is ideal when it is difficult to define the selection required but easy to define a selection of entities that are not wanted.

To use this command, click **Edit ► Invert Selection** or right-click and select **Invert Selection** from the shortcut menu.

DeSelect All

Use the Deselect All command to clear the current selection of entities. To cancel any selection so no entities are highlighted, click **Edit ► Deselect All**. You can also right-click on the selection, and choose **Deselect** from the shortcut menu.

Edit Properties

To use the Properties Alt+Enter command, do the following:

- Click **Edit ► Edit Properties** or press **Alt+Enter**. The *Edit Group Properties* dialog box appears (Figure 6-20).
- Click on the following tabs to edit the Point, Line, Arc, Circle, String, Polygon, Text, Dimension, and Lot properties.

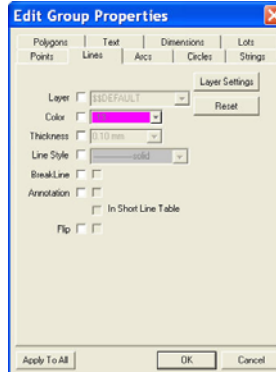



Figure 6-20. Edit Group Properties

Repeat

Use the Repeat command as a shortcut to repeat the action or routine that was just executed.

To use the last command again, click **Edit ▶ Repeat** or click the **Repeat**  icon on the **Standard** toolbar. You can also right-click on the project area, and select **Repeat** from the shortcut menu.

Survey View: View Menu

This section describes the different **Survey View** menu commands on the View menu located on the main menu bar (Figure 7-1).

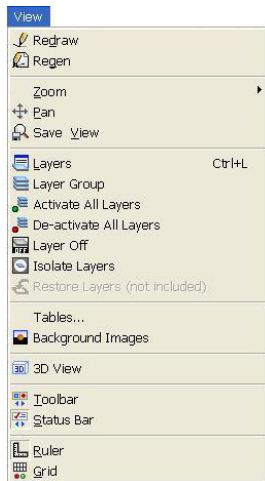


Figure 7-1. Survey View – View Menu

In This Section:


- “Redraw” on page 7-2
- “Regen” on page 7-2
- “Zoom” on page 7-2
- “Pan” on page 7-5
- “Layers” on page 7-7
- “3D Viewer” on page 7-15

- “Sets the value of the vertical exaggeration of 3D View. This is useful when flat surfaces display with very little relief. Setting this value to a higher number will show the relief in an exaggerated view.settingToolbar” on page 7-19
- “Status Bar Options” on page 7-22
- “Ruler” on page 7-23
- “Grid” on page 7-23

Redraw

Click **View ▶ Redraw** or click the **Redraw**  icon on the **Standard** toolbar to refresh the display of entities in the current window.

Regen

Click **View ▶ Regen** or click the **Regen**  icon on the **Standard** toolbar to regenerate the display of data by reading the data from the project database.

Zoom

Use the Zoom command to remap the data on the screen at a different scale.

You can access the **Zoom** commands (Figure 7-2 on page 7-3) in the following ways:

- click on the **View** menu
- Right-click shortcut menu
- Icons
- Keyboard
- Scroll wheel on the mouse.

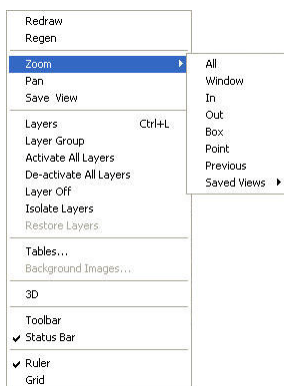






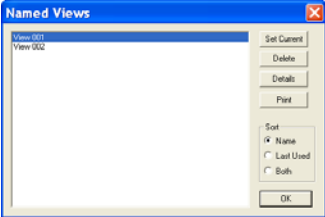
Figure 7-2. Zoom Option and Submenu

Table 7-1 describes the Zoom sub-menu commands.

Table 7-1. Zoom Command Descriptions

Zoom Submenu	Standard Toolbar Icon	Description
All		Data is displayed with all of the entities in the project fitting into the display area.
Window		Click and drag the zoom window around a specific area you want to magnify. Only the data in the zoom window is displayed.
In		The data in the center of the view is magnified, and data at the display edges will disappear from view. Moving the scroll wheel on the mouse away from you also magnifies the view at the position of the pointer.
Out		The data in the center of the view is reduced, and data currently off the screen may appear. Moving the scroll wheel on the mouse toward you also reduces the view at the position of the pointer.

Table 7-1. Zoom Command Descriptions (Continued)

Zoom Submenu	Standard Toolbar Icon	Description
Box		<p>A yellow box appears on the screen. Position the center of the box at the zoom point and click. The data in the box is mapped to the full extents of the display area. Right-click, and select Cancel to finish.</p>
Point		<p>This centers the display on a selected point or defined coordinates at a defined scale. The input bar displays on the lower left of the display area. First, enter a display scale in the Scale field, and then enter the required point number or Easting and Northing values. The display zooms to the defined scale with the defined point in the center.</p>
Previous		<p>The previous zoomed view is displayed.</p>
Saved Views (Named Views)		<p>Click to access the <i>Named Views</i> dialog box.</p>  <p>Do the following to edit the Named View(s):</p> <ul style="list-style-type: none"> • Current – sets the highlighted view as one currently being used. • Delete – deletes the highlighted view. • Details – gives details about the highlighted view. • Print – prints out the highlighted view. • Sort Panel – sorts the named views either by name, when it was last used, or both. • OK – saves and exits the Names Views dialog box.




To zoom to the extents of all entities currently displayed on active layers, double-click the scroll wheel of the mouse.

Pan

Use the Pan command to move the data position in the display area without changing its mapped scale.

To move the data from one position to another, do the following:

1. Click **View ▶ Pan** or click the **Pan**  icon on the **Standard** toolbar.
2. Click to define the first position.
3. Click to define the second position (the direction in which you want to move the view).

This moves the first point to the relative position of the second point.

4. Right-click, and select **Cancel** from the shortcut menu to finish the pan option.



Press and hold the scroll wheel on the mouse at a chosen position and drag the pointer to a new position.

Save View

The Save View command is useful if you have to zoom to different parts of the display area frequently. Named views can be saved for each area that you zoom to. You can also print and restore each view.

1. Click **View ▶ Save View** to save the current view displayed on the screen as a named view.

The *New View Name* dialog box appears (Figure 7-3 on page 7-6).

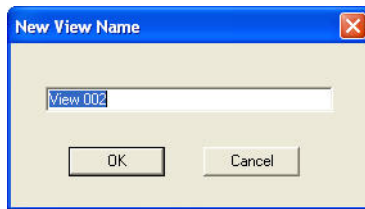


Figure 7-3. New View Name

2. Enter a name or accept the default name, and click **OK**.
3. To zoom to a saved view, right-click and select the name of the saved view from the **Saved Views** submenu.
4. To print, delete, sort, set as current, or display the details of a saved view, right-click and select **Named Views** from the submenu.

The *Named Views* dialog box appears (Figure 7-4), listing the saved views.

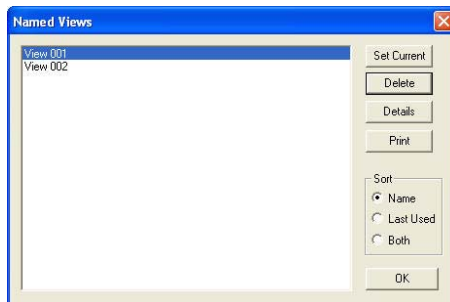


Figure 7-4. Named Views Window

5. Select a saved view, and click an option on the right side of the window.

Toolbar Settings

When an entity is created, the individual properties of that entity can be varied from the default settings using the **Properties** toolbar. Once a change has been made in the toolbar, the next entity and further entities will be created with those changed properties. The new properties selected on the **Properties** toolbar (Figure 7-5) become the default properties for the creation of further entities until they are changed. This will not alter the properties of entities created before the change.




Figure 7-5. To and From Arrows in the Properties Toolbar

Click the right arrow on the **Properties** toolbar to copy the properties set for the current layer in the *Layer Settings* dialog box to the **Properties** toolbar. You can also click the left arrow to copy the properties displayed for the current layer on the **Properties** toolbar as default settings for the layer in the *Layer Settings* dialog box.

This establishes standard settings for a layer and sets the properties on the **Properties** toolbar from the layer defaults to ensure standards for all properties on a specific layer.

Layers

Use the Layers command to create a new layer, rename a layer, edit the properties of existing layers, delete layers, or control the display of layers. Think of layers as individual sheets of clear film that can be viewed one at a time or combined in a stack. Each entity belongs to one layer only and may be activated or deactivated by turning its layer on or off. Several layers may be activated or deactivated together.

Click **View ► Layers** to open the *Layer Settings* dialog box (Figure 7-6 on page 7-8) or click the **Layer Settings**  icon on the toolbar.

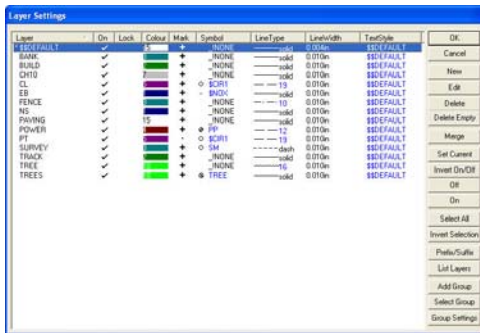


Figure 7-6. Layer Settings

All new projects will have one layer named **\$DEFAULT**.

Only one layer can be current at a time. The current layer is marked with an asterisk *. Data entered into the project using the insert entity options will be created on the current layer.

Columns The properties for each layer are arranged in columns. The title of each column describes the property displayed in that column. See Table 7-2 for a description of each column.

Table 7-2. Layer Settings Descriptions

Column Name	Column Description
Layer	Name of each layer
On	A check mark indicates the layer is on
Lock	A check mark indicates the layer is off
Color	The color defined for this layer
Mark	The point mark defined for this layer
Symbol	The symbol defined for this layer
Line Type	The line style defined for this layer
LineWidth	The line thickness defined for this layer
TextStyle	The text style defined for this layer.

Click the title of the column to list the layers in order of their properties in that column. The layers are set from the feature coding in the dataset and the properties set for each feature code in the Survey Code Library.

Selecting Layers You can use the mouse, arrow keys, or the **Ctrl** and **Shift** keys to select one or multiple layers. The selected layers are highlighted. You can also use the **Select All**, **Invert Selection**, or **Set Current** buttons to make a selection. See Table 7-3 for more information.

Command Buttons There are several command button options on the right side of the *Layer Settings* dialog box to control the layers. For a description of the buttons, see Table 7-3 below.

Table 7-3. Button Descriptions

Button Name	Button Description
OK	Accepts the new settings and returns to the Survey View.
Cancel	Closes the dialog box.
New	Adds a new layer.
Edit	Edits the properties of an existing layer.
Delete	Deletes the selected layer.
Delete Empty	Deletes layers that do not contain any entities.
Merge	Combines all selected layers into a single specified layer.
Set Current	Sets the current layer. Select one layer and click this button. The current layer is marked with an asterisk * in the list. The current layer displays on the Properties toolbar.
Invert On/Off	Switches layers that are off to the on status and layers that are on to the off status.
Off	Removes the check mark in the On column next to the selected layer.
On	Places a check mark in the On column next to the selected layer.
Select All	Selects all of the layers in the dialog box
Invert Selection	The unselected layers become selected and the currently selected layers become unselected.
Prefix/Suffix	Adds a prefix or suffix to the existing layer name.
List Layers	Generates a printable report of specified layers.
Add Group	Allocates a selection of layers to a named group, so they can be turned on or off together. A layer can belong to more than one group.
Select Group	Automatically selects the layers that belong to a group or groups.

Table 7-3. Button Descriptions (Continued)

Button Name	Button Description
Group Settings	Modifies a layer group.

Add Layers

To add a new layer, do the following:

1. Click **View ▶ Layers**. The *Layer Settings* dialog box appears. See Figure 7-6 on page 7-8.
2. Click **New** on the *Layer Settings* dialog box.
3. The *Add New Layer* dialog box appears (Figure 7-7) with a default layer name and layer properties.

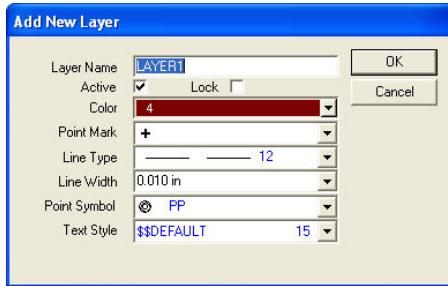


Figure 7-7. Add New Layer

4. Type a name for the new layer. Enter a maximum of D1 alphanumeric characters.
5. To turn the layer on, check mark the **Active** box.
6. To lock the new layer, check mark the **Lock** box.
7. Select all other properties from the appropriate drop-down menus. This is optional but can be used if all entities on a layer have the same properties.
8. Click **OK** to accept the settings and return to the *Layer Settings* dialog box (Figure 7-6 on page 7-8). The new layer will be set as the current layer.

Edit Layers

To edit the properties of an existing layer or to rename a layer, do the following:

1. Click **View ► Layers**. The *Layer Settings* dialog box appears. See Figure 7-6 on page 7-8.
2. Select the desired layer, and then click **Edit** on the *Layer Settings* dialog box.

The *Edit Survey Layer* dialog box appears (Figure 7-8), displaying the layer's current properties.

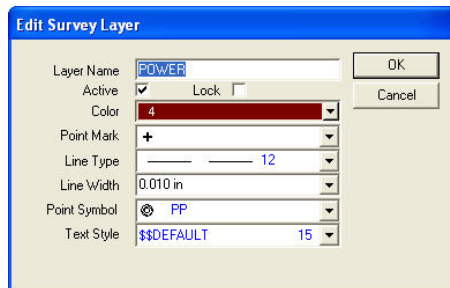


Figure 7-8. Edit Survey Layer

3. Make the appropriate edits, and then click **OK** to accept the changes and return to the *Layer Settings* dialog box.

Editing Multiple Layers To edit the properties of multiple layers, do the following:

1. Click **View ► Layers**. The *Layer Settings* dialog box appears. See Figure 7-6 on page 7-8.
2. Select the desired layers, and then click **Edit** on the *Layer Settings* dialog box. Make appropriate edits.

The *Edit Survey Layers* dialog box appears (Figure 7-9 on page 7-12). The *Layer Name* field is set to MULTIPLE_EDIT to indicate that several layers are being edited.

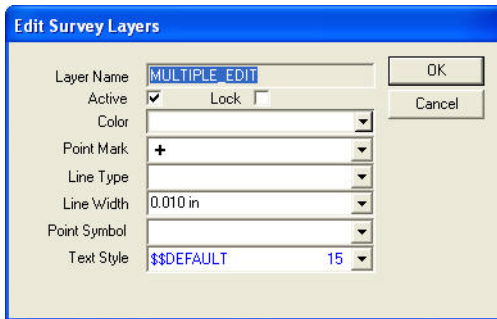


Figure 7-9. Editing Multiple Layers

3. Click **OK** to accept the changes and return to the *Layer Settings* dialog box.

Deleting a Layer To delete a layer or multiple layers, do the following:

1. Click **View** ► **Layers**. The *Layer Settings* dialog box appears (Figure 7-6 on page 7-8).
2. Select the desired layer(s), and click **Delete** on the *Layer Settings* dialog box.

A confirmation dialog box appears (Figure 7-10).



Figure 7-10. Delete Confirmation Window

3. Click **Yes** to delete the layer(s) or **No** to cancel the operation.

Adding a Group To create a group of layers, do the following:

1. Click **View** ► **Layers**. The *Layer Settings* dialog box appears. See Figure 7-6 on page 7-8.

- Highlight the layers for the group, and click **Add Group** on the *Layer Settings* dialog box (Figure 7-11).

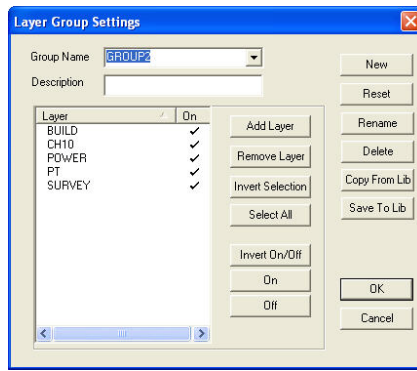


Figure 7-11. Layer Group Settings

- Type a group name and description in the appropriate fields.
- In the **On** column, each layer has a check mark by default to indicate it is active in the group. If you do not want a layer to be active, highlight the layer to deselect it.

Control Layers

The command buttons on the *Layer Group Settings* dialog box performs the following operations for layers (Figure 7-11):

- **Add Layer** – adds a layer to the group. When pressed, the Layer List window appears. Select the layer(s) from this list or type the name of a layer to be added to the group.
- **Remove Layer** – removes the selected layer(s) from the group.
- **Invert Selection** – selects the layers that were not previously selected. The layers that were selected become unselected.
- **Select All** – selects all of the layers.
- **Invert On/Off** – toggles between activating a layer and deactivating it by placing a check mark in the **On** column and removing it.
- **On** – places a check mark next to a selected layer indicating it is active.

- **Off** – removes a check mark next to a selected layer indicating it is not active.
- **Reset** – resets the settings for a selected layer group to previously saved settings prior to editing.
- **Rename** – renames a selected layer group.
- **Copy From Lib** – copies the layer group settings from the MAGNET Office library into the current job.
- **Save To Lib** – saves the layer group settings to the MAGNET Office library from the current project.
- **OK** – accepts the settings and returns to the *Layer Settings* dialog box.
- **Cancel** – closes the dialog box.

Selecting a Group To automatically select the layers that belong to a group or groups:

1. Click **Select Group** on the *Layer Settings* dialog box (Figure 7-6 on page 7-8).

The *Layer Group Selection* dialog box appears (Figure 7-12).

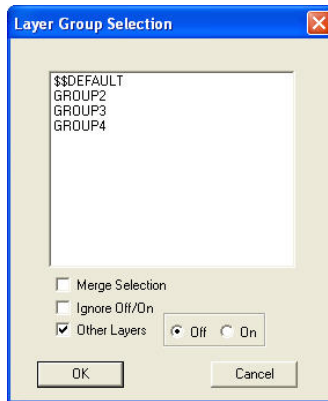


Figure 7-12. Layer Group Selection

2. On the *Layer Group Selection* dialog box, make the following selections:
 - **Merge Selection** – creates a single list of selected layers if more than one group is selected (when check marked).

- Ignore Off/On – toggles between ignoring the On/Off flag defined for each layer within a group or not (when check marked).



The **Layer Settings** dialog box automatically highlights all of the layers in the chosen groups, and the On/Off flags are set, depending on the options you selected.

- Other Layers – sets the On/Off flag for other layers in the project that are not related to any group in the selection.
- Off/On – select the Off/On option as required.
- **OK** – accepts/saves the settings.
- **Cancel** – cancels your selection/ quits the routine.

3D Viewer

On the *3D Viewer* screen, click on the **View** menu option to set the required parameters for the view.

Click **View** ► **3D View**. The *3D View* dialog box appears (Figure 7-13 on page 7-16). Enter or select the following parameters:

- Select From – enable one of the following parameters to display in the 3D Viewer:
 - All Data: enable to display all drawing data in the 3D Viewer, whether the layer visibility is ON or OFF.
In addition, the transfer line work option is required to be toggled on in order to display the linework in the 3D Viewer.
 - Active Layers: enable this option to only display the data contained in the active layers within the 3D Viewer.
In order to display linework, the transfer linework option should be toggled ON.
 - Current Selection: enable this option to only display the entities within the current selection.
In order to display linework, the transfer linework option should be toggled ON.

- Driver Offset – use this option for the inspection of the surface at a specified perpendicular offset, either *Left* or *Right* of the alignment.
- Alignment Spacing – enter a value to determine the smoothness of the curvature in an alignment. The lower the value, the smoother the curvature will be.

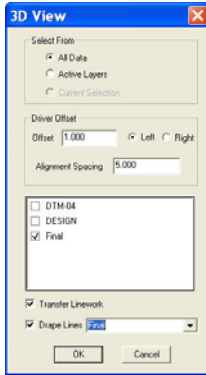


Figure 7-13. Multiple DTM Display Example

- DTM Window – this window displays all available DTMs in the project (Figure 7-14).

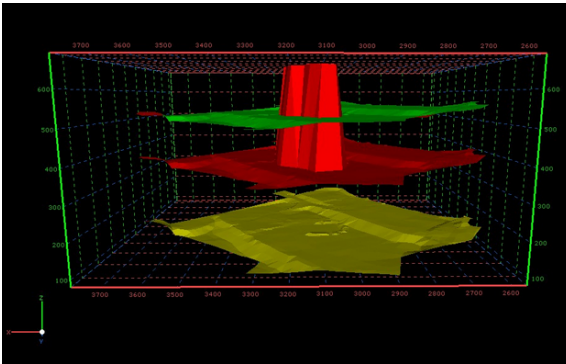


Figure 7-14. Example of Multiple DTMs on Display

- Transfer Linework – check mark this box to transfer specific linework into the 3D Viewer.
- Drape Lines – check mark this box to drape all selected lines onto the selected surface in the 3D Viewer.

- **OK** – press to save. The *3D Viewer* dialog box appears (Figure 7-15).

Settings Menu

The Settings menu on the *3D Viewer* screen (Figure 7-15) has the following options:

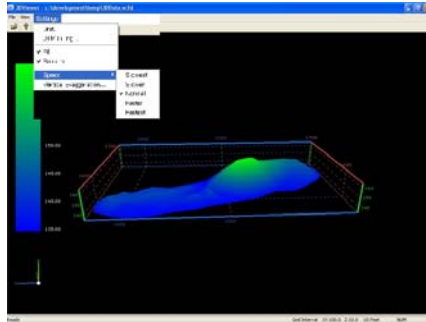


Figure 7-15. 3D Viewer – Settings Menu

Unit

1. Click **Settings** ► **Unit**. The *Unit* dialog box appears (Figure 7-16).
2. Select either a unit of measurement, either *Metric*, *US feet*, or *International feet*.
3. Click **OK** to return to the main screen.



Figure 7-16. Unit of Measurement

With the grid displayed, when the units are changed, the grid will be updated to display the selected units.

DTM Filling

The DTM Filling option controls how the surface displays.

1. Click **Settings** ▶ **DTM Filling**.

The *DTM Fill* dialog box appears (Figure 7-17).

2. Select one of the *Shading* options to display the surface:

- Based on light source – this is the legacy option that displays the colored fill of the surface model triangles..
- Based on elevations – enable this option to provide a color of the DTM, based upon the lowest and highest elevations.
 - *Color of lowest elevation*: sets the color for the lowest elevation in the surface.
 - *Color of highest elevation*: sets the color for the highest elevation in the surface.

3. Press **OK** to save and return to the *3D Viewer* screen (Figure 7-15 on page 7-17).

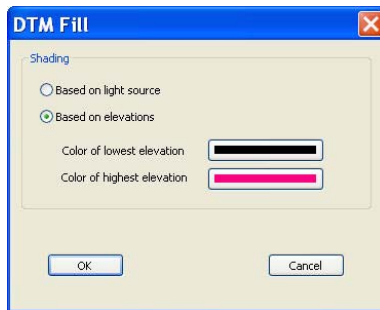


Figure 7-17. DTM Fill Options to Display the Surface

Fill

Toggle on and off to control the filling of the DTM mesh.

- Toggle ON – mesh will be filled, based upon the DTM filling settings.
- Toggle OFF – only the mesh will display.

Smooth

Smooths the triangular mesh lines to provide a smoother appearance. The smoothing for the color transition of the mesh is also controlled with this option. Leave this setting ON to obtain the best visual affect for the surface.

Speed

This option controls the speed of zooming and panning while in 3D view. In addition, when following an alignment along the surface, this setting controls the speed of the “drive”. Select either *Slower*, *Slowest*, *Normal*, *Fastest*, or *Faster*.

The speed setting is also available on the context “pop-up” menu. Do a right-click any time while in the 3D Viewer.

Vertical Exaggeration

Sets the value of the vertical exaggeration of 3D View. This is useful when flat surfaces display with very little relief. Setting this value to a higher number will show the relief in an exaggerated view.
view.settingToolbar

Use the Toolbar command to control the display of toolbars in the Survey View. Upon first opening MAGNET Office, the toolbars are stacked at the top of the screen. You can show or hide the toolbars, separate them, and reposition them anywhere in the window vertically and horizontally by using the click and drag method.

To show or hide a toolbar, do the following:

- Click **View ▶ Toolbar**.

The *Configure Toolbars* dialog box appears (Figure 7-18 on page 7-20).

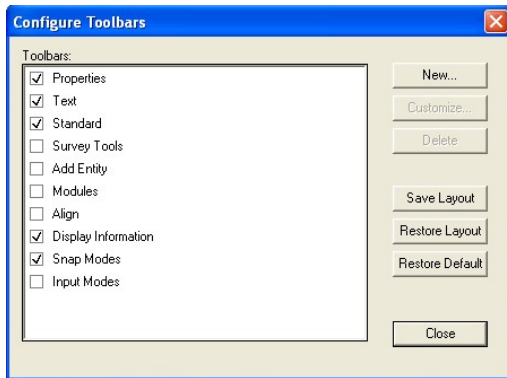


Figure 7-18. Configure Toolbars

On the *Configure Toolbars* dialog box (Figure 7-18), make the following selections:

- **New** – press to name a toolbar on the *New Toolbar* dialog box (Figure 7-19).

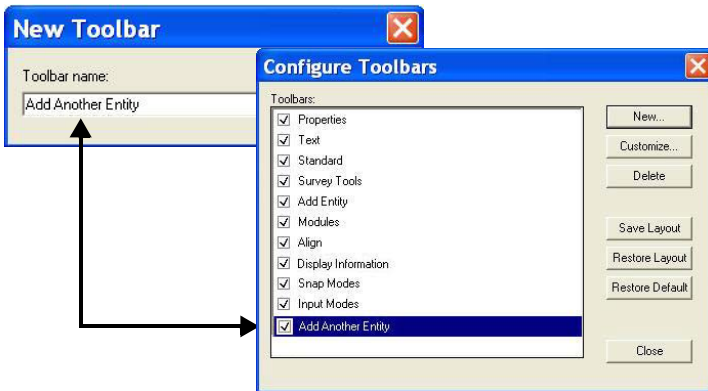


Figure 7-19. Configuring Toolbars

- **Customize** – press to customize the newly created toolbar (Figure 7-20 on page 7-21), then do the following:
 - Available toolbar buttons: highlight a toolbar in this column, then click the right arrows to move to the next column.
 - Current toolbar buttons: toolbars from the Available toolbar buttons column displays here.

- **Move Up/Down:** press **Move Up** to move a toolbar selection up or **Move Down** to move a toolbar selection down.
- **Sep After:** press to leave a space between toolbar selections.
- **Restore:** press to bring back a toolbar previously removed from the column.
- **Close:** press to exit the *Configure Toolbars* dialog box.

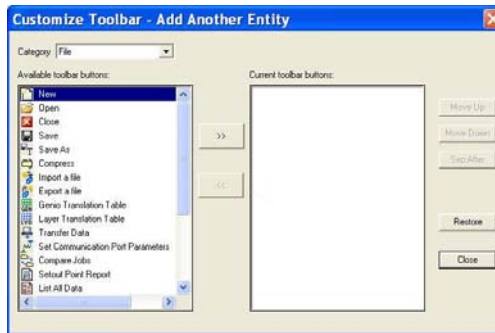


Figure 7-20. Customize the New Toolbar

- **Delete** – press to delete a highlighted toolbar.
- **Save Layout** – press to save the position of the toolbars.
- **Restore Layout** – press to restore the layout previously deleted.
- **Restore Default** – press to restore the toolbars to their default positions in every view.
- **Close** – press to return to the main screen.



To hide a toolbar, clear the check mark from the box beside it. The check mark indicates the toolbar is displayed on the screen.

You can arrange the toolbars inside or outside of the project area by clicking a toolbar (not an icon) and dragging to where you want to position it (Figure 7-21 on page 7-22). Once you have moved a toolbar into the working area, you can also change the shape of the toolbar to fit more appropriately by placing the pointer over an edge of the toolbar until it changes to double arrows, and then clicking and dragging the toolbar edge (Figure 7-21 on page 7-22).

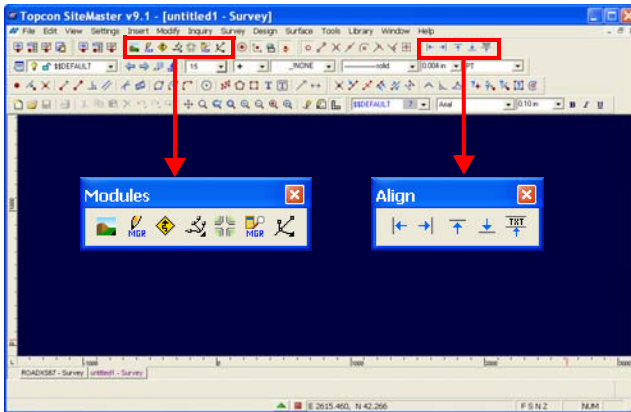


Figure 7-21. Positioning Toolbars

The names of the individuals tools on the toolbar are displayed when the pointer is moved over an icon.

Change Views

You can move between the Survey View, Drawing View, Road View, and Raw Data Editor by clicking the icons on the **Modules** toolbar (Figure 7-22).

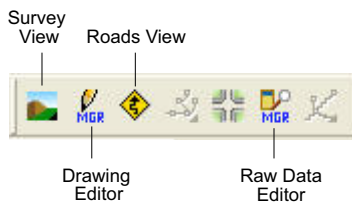


Figure 7-22. Modules Toolbar

Status Bar Options

The status bar on the lower left of the display area provides information and several options. The Status Bar displays by default. The check mark next to Status Bar in the **View** menu indicates the status bar will be displayed. To hide the Status Bar, click to clear the

check mark. For a description of the Status Bar, see Figure 7-23 and Table 7-4 below.




Figure 7-23. Status Bar

Table 7-4. Status Bar Description

Number	Description
1	The green triangle provides quick access to the DTM Display Settings.
2	The red rectangle, also called the Hot Button , opens the General Settings dialog box that control the display of entities and their attributes on the screen.
3	The rolling coordinates display the coordinates of the pointer position.
4	This provides quick access to the project unit settings.
5	This indicates the Number Lock is activated.

Ruler

The ruler appears at the edge of the display area. A red line on the ruler indicates the coordinate of the pointer position. The scale and division of the rulers depend on the mapping of the data in the Survey View.

To display or hide the Ruler, click **View ► Ruler** or click the **Ruler**  icon on the Standard toolbar.


Grid

Click **View ► Grid** to display the grid in Survey View. Click again to hide the grid.

Correct Positioning Data Using the Grid Command

Use the Grid command to help correct data positioning, especially if the **Snap Grid Point** command will be used. A grid may be defined with major and minor spacings with suitable colors and line styles.

To define Grid settings, do the following:

1. Click the **Hot Button**  on the Status Bar on the lower left of the display area.

The *General Settings* dialog box appears (Figure 7-24).

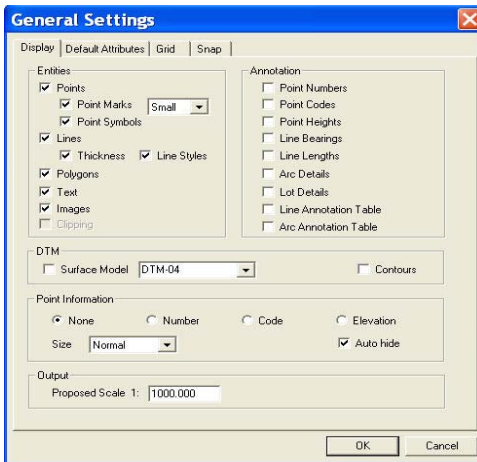


Figure 7-24. General Settings – Grid Tab

2. Set the following parameters on the *Grid* tab:
 - Spacing X and Y – type the X and Y spacing for the grid. These are the major grid spacings.
 - Divisions X and Y – the grid can be divided into smaller sections. Type the X and Y division number. For example: typing in 5 will divide the major grid into 5 minor divisions. If the major grid distance is X=100 and Y=100, then the minor grid will be X=20 and Y=20. If the divisions are set to 1, then only the major grid spacings are displayed.
 - Color: Select the color for both major and minor grids.

- Origin: Type the East and West origin for the grid. This is usually set to $E=0.00$ and $W=0.00$. Changing the origin only alters the grid position; it does not alter the coordinate values of the points.
 - Minimum Display Interval (pixels): Type a pixel value to ensure the grid is not drawn too densely when viewing a large section of the project. If the grid spacing is less than the minimum pixel interval, then the small grid sections will not be displayed.
 - Show Grid – check mark this box to display the grid in the Survey View (Figure 7-25).
3. Click **OK**.

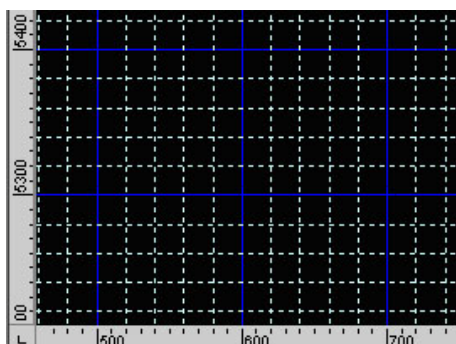


Figure 7-25. Show Grid

Notes:

Survey View: Settings Menu

This section describes the different Survey View menu commands on the Settings menu located on the main menu bar (Figure 8-1).

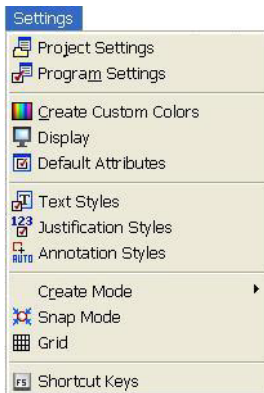


Figure 8-1. Survey Menu – Settings Menu

In This Section:

- “Project Settings” on page 8-2
- “Program Settings” on page 8-10
- “Create Custom Colors” on page 8-22
- “Display General Settings” on page 8-23
- “Default Attributes” on page 8-26
- “Text Styles” on page 8-28
- “Justification Styles” on page 8-31
- “Define Annotation Styles” on page 8-35
- “Point Creation Mode” on page 8-53

- “Apply the Snap Mode” on page 8-57
- “Shortcut Keys” on page 8-60

Project Settings

Use the *Project Settings* dialog box to define settings, record information, and maintain statistics for a particular project. Project settings apply only to the open project. Default settings defined in the program settings are automatically applied to each new project when it is created.

To define project settings, do the following:

1. Click **Settings** ► **Project Settings**.

The *Project Settings* dialog box appears (Figure 8-2).



Figure 8-2. Project Settings Dialog Box

Click each tab to access different settings and modify them as necessary.

2. Click **Print** to print the current project settings to a report. The report will display in a new dialog box and may be printed or saved to a report (.rep) file.

For a description of each tab, see the following:

- “Project Details” on page 8-3
- “Project Changes” on page 8-4

- “Statistics” on page 8-6
- “Unit Settings” on page 8-7
- “Symbols” on page 8-8
- “Coordinate Systems” on page 8-9

Project Details

Use the *Project Details* dialog box to view and enter user, company, and project details. User and company information automatically appears if you already entered and saved this information into the program settings.

To enter user and project details, do the following:

1. Click **Settings ▶ Project Settings**.
2. On the *Project Settings* dialog box (Figure 8-2 on page 8-2), press on the *Project Details* tab (Figure 8-3) to complete each field. For field descriptions, see Table 8-1 on page 8-4.

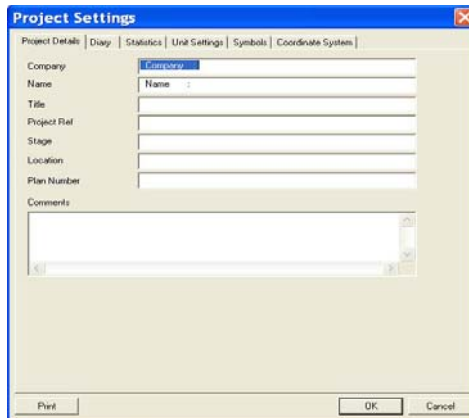


Figure 8-3. Project Settings – Project Details Tab

- When you have finished, click **OK** to exit the *Project Settings* dialog box or click on another tab.

Table 8-1. Project Details – Field Descriptions

Field	Description
Company Name	Enter the name or edit as required.
Name	Enter the name or edit as required.
Title	Enter a project title.
Details 1-4	Enter details for this project. The headings for Details 1-4 may have been modified using the User Information tab in the Program Settings.
Comments	Enter any comments to assist with project control.

Project Changes

Use the *Diary* tab to enter project information to keep track of time spent and changes made to a project. You can add new entries as work progresses and MAGNET Office will automatically calculate the total time.

To make diary entries, do the following:

- Click **Settings** ► **Project Settings**. The *Project Settings* dialog box appears (Figure 8-4 on page 8-5).
- Click on the **Diary** tab, then click **New**. The *New Diary Entry* dialog box appears (Figure 8-4 on page 8-5).

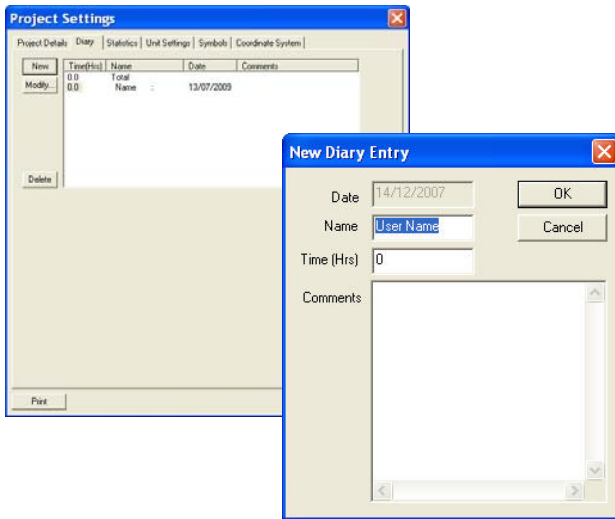


Figure 8-4. Project Settings – Diary Tab

The current date set in Microsoft Windows is automatically entered in the *New Diary Entry* dialog box (Figure 8-4).

3. Enter the time spent on this project and any comments about the work completed during this time.
4. To delete an entry, select it and click **Delete**.
5. When you have finished, click **OK** to exit the *Project Settings* dialog box or click on another tab.

The entry is added to the main list for this project on the **Diary** page. Each new entry may be modified.

To modify a diary entry, do the following:

1. On the *Diary* tab, select an entry, and press **Modify**. The *Modify Diary Entry* dialog box appears (Figure 8-5 on page 8-6).

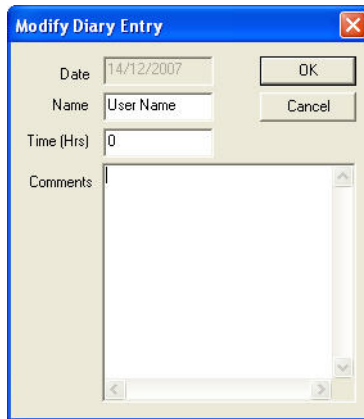


Figure 8-5. Diary Tab – Modify Diary Entry

2. Enter the revised details, and click **OK** to save your edits and close the window.
3. Click **OK** to exit the *Project Settings* dialog box or click another tab.

Statistics

Use the *Statistics* tab on the *Project Settings* dialog box to view the current project statistics (Figure 8-6 on page 8-7):

1. Click **Settings** ▶ **Project Settings** ▶ **Statistics**.

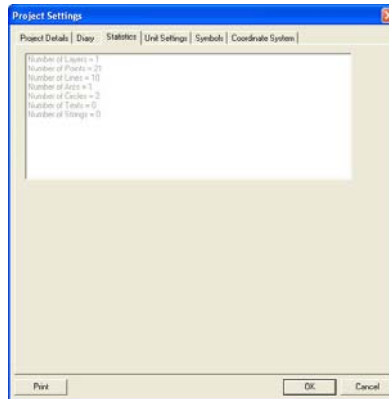


Figure 8-6. Project Settings – Statistics Tab

2. Click **OK** to exit the *Project Settings* dialog box or click another tab.

Unit Settings

Use the *Unit Settings* tab on the *Project Settings* dialog box to view and set the units of measurement and other parameters required for data entry and computation.

1. Click **Settings ▶ Project Settings**.
2. On the **Unit Settings** tab (Figure 8-7 on page 8-8), modify each field as necessary. For field descriptions, see Table 8-2 on page 8-15.

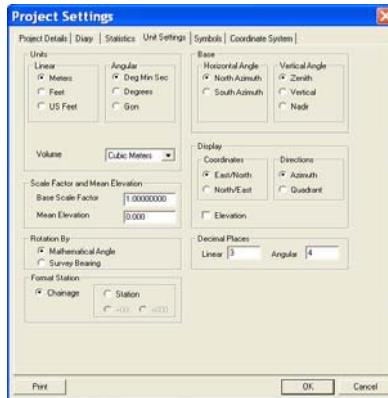


Figure 8-7. Program Settings – Unit Settings Tab

These settings are initially taken from the settings defined in the *Program Settings* menu option.

3. When you have finished, click **OK** to exit the *Project Settings* or click on another tab.

Symbols

Use the *Symbols* tab on the *Project Settings* dialog box to view the default characters for certain symbols in the current project. These settings are initially taken from those defined in the *Program Settings* menu option. You can modify these settings on a project-by-project basis.

To set symbols parameters, do the following:

1. Click **Settings** ► **Project Settings**.
2. On the *Symbols* tab (Figure 8-8 on page 8-9), modify each field as necessary.

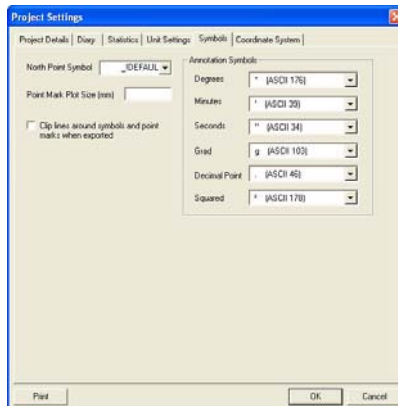


Figure 8-8. Project Settings – Symbols Tab

3. Click **OK** to exit the *Project Settings* dialog box or click another tab.

Coordinate Systems

To work in a predefined coordinate system, set the following parameters on the *Coordinate System* tab to (Figure 8-9 on page 8-10):

- **Select projection** – Specify the coordinate system from the list of provided systems in the **Select projection** pane.
- The parameters for the selected system display in the **Selected parameters** pane on the right.
- **Projection ellipsoid** – Use the default projection ellipsoid or select one from the drop-down list.
- **Geoid file** – Select a geoid file to use in conjunction with your defined settings, if needed.

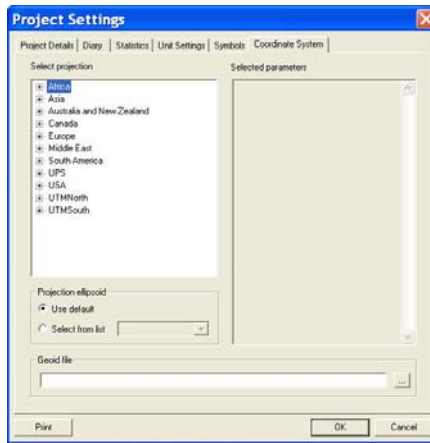


Figure 8-9. Project Settings – Coordinate System Tab

Program Settings

When installing MAGNET Office, define the program settings for the standard interface. These settings are global and will be the default settings for any new projects that you create. You can have different settings for various clients or project type by saving the settings to a file and loading them into a new project at a later time.



You can edit these settings on a local file-by-file basis using the menu options on the *Project Settings* dialog box in the **Settings** menu.

To define program settings, do the following:

1. Click **Settings** ► **Program Settings**.

The *Program Settings* dialog box appears (Figure 8-10 on page 8-11).



Figure 8-10. Program Settings – User Information Tab

Click on each tab to access different settings and modify them as necessary.

For a description of each tab, see the following:

- “User Information”
- “Folders” on page 8-12
- “Libraries” on page 8-13
- “Unit Settings” on page 8-14
- “Symbols” on page 8-17
- “Coordinate System” on page 8-18
- “Preferences” on page 8-19

User Information

To enter company information and to define labels, click on the *User Information* tab (Figure 8-11 on page 8-12).

1. Click **Settings** ▶ **Program Settings**. The *Program Setup* dialog box displays (Figure 8-11 on page 8-12).



Figure 8-11. Program Settings – User Information Tab

2. Enter identification information in each field, and then select the next tab to continue the program setup.
3. Click **OK** to exit the *Program Settings* dialog and save your changes or click on another tab.

Folders

When data and libraries are shared on a file server, use an appropriate path and folder name for each setting. You can change the settings by highlighting an item, then clicking **Browse** to select the appropriate path on the *Folder* tab on the *Program Settings* dialog box (Figure 8-12 on page 8-13).

MAGNET Office automatically follows these paths and searches these locations to open and save files.

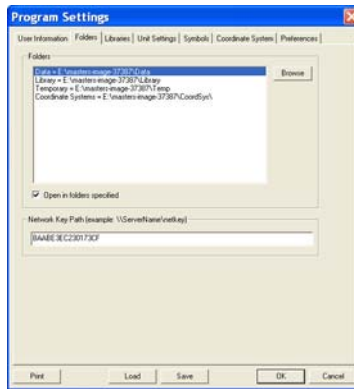


Figure 8-12. Program Settings – Folder Tab

The default directory for the MAGNET Office installation is *c:\MAGNET Office*, and the default paths for the settings in this option are set as subfolders in the program folder.

The paths set here are automatically followed if a routine is opening or if you are saving data into one of these folders. These folders may be stored on a file server or a shared computer and accessed across a network; however, the program files for MAGNET Office must be loaded on the local hard drive of the work station.

Libraries

Library files store regularly used information, such as symbols, line styles, and survey feature codes. These files need to be identified and the paths to their locations mapped. These paths should be checked and established when MAGNET Office is first installed; however, you may need to access other library files in different locations for a specific job.

Click **Settings** ► **Program Settings** and click on the *Libraries* tab on the *Program Settings* dialog box. The libraries that display on this tab are opened when a job is started. Users should check that all libraries are suitable for the work being performed.

All of the library files needed for running MAGNET Office are listed (Figure 8-13 on page 8-14).

To change these settings, highlight a library file, and click **Browse** to search for another file. The path shown on the **Browse** option is also shown on the *Folder* tab.

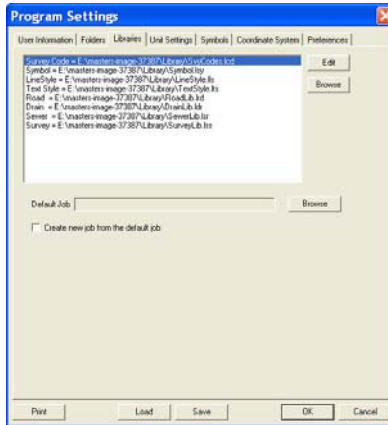


Figure 8-13. Program Settings – Libraries Tab

Unit Settings

The *Unit Settings* tab establishes the settings for calculation and display of measurement units and becomes the default settings for all new projects.

Click **Settings** ► **Program Settings** to change the settings of individual projects. See Figure 8-14 and Table 8-2 on page 8-15.



Figure 8-14. Program Settings – Unit Settings Tab
 Table 8-2. Unit Settings – Field Descriptions

Field Name	Field Description
Units	
Linear	<p>Meters: International standard unit of one meter (default)</p> <p>Feet: International measure of one foot being 0.3048 meters</p> <p>US Feet: U.S. measure of one foot being 0.3048006 meters or 39.37 inches to the meter</p>
Angular	<p>Deg Min Sec: Degrees, minutes, and seconds (default). Data is entered in a decimal format with Degrees before the decimal point, then the first two digits after the decimal are Minutes, and the third and fourth digits are Seconds. This format displays as DDD.MMSS. Seconds can be taken to decimal values if the number of decimal places for Angular Measure is increased. For example: at 6 decimal places, a value such as 30.042235 represents 30 degrees, 4 minutes, and 22.35 seconds.</p> <p>Decimal Degrees: Units are degrees only and minute and second values are expressed as a decimal of the degree.</p> <p>Gon: The circle is divided into 400 gons, so 100 gons is equal to 90 degrees.</p>
Volume	Select either cubic feet for cubic yards for the volume calculation report.

Table 8-2. Unit Settings – Field Descriptions (Continued)

Field Name	Field Description (Continued)
Scale Factor and Mean Elevation	
	<p>Base Scale Factor: Defines the Projection Scale Factor used as the default setting for all new projects. The default is 1.00000 and will be the normal setting for users.</p> <p>If variable Projection Scale Factors are used, then it is advisable to set these individually for each project on the Unit Settings tab of the Project Settings dialog box.</p> <p>The Scale Factor is applied to all distances when coordinate positions are computed. The ground distance will always be shown when computed lines or missing lines are enquired.</p>
	<p>Mean Elevation: Sets the Mean Elevation for all projects. The default is 0.000. If a sea-level correction is applied to distances, it is advisable to set the mean elevation individually for each project on the Unit Settings tab of the Project Settings dialog box.</p> <p>The sea-level correction is applied to all distances when coordinate positions are computed. The ground distance will always be shown when computed lines or missing lines are enquired.</p>
Rotation By	
	<p>Mathematical Angle: Defines the orientation in standard CAD mode (east is 0.0000 and north is 90.0000).</p> <p>Survey Bearing: Defines the orientation of text in bearing mode, similar to the mode used for lines (north is 0.0000 and east is 90.0000).</p>
Format Station	
	<p>Select either Chainage or Station as a format.</p>
Base	
Horizontal Angle	<p>North Azimuth: The horizontal angle is defined with zero degrees set to the north.</p> <p>South Azimuth: The horizontal angle is defined with zero degrees set to the south.</p>
Vertical Angle	<p>Zenith: 90° is level.</p> <p>Vertical: Zero degree is level.</p> <p>Nadir: The vertical angle is set with the zero at the Nadir, which is immediately below the observer or 180 degrees from the zenith with the horizon at 270 degrees.</p>
Display	

Table 8-2. Unit Settings – Field Descriptions (Continued)

Field Name	Field Description (Continued)
Coordinates	<p>East/North: The input and output of coordinate values will show the east value followed by the north value.</p> <p>North/East: The input and output of coordinate values will show the north value followed by the east value.</p>
Bearings	<p>Select one of the following:</p> <ul style="list-style-type: none"> • WCB (0-360) (displays direction in azimuth) • Quadrant: directions of north and south are deviated to the east or west. A bearing of 10 degrees, 35 minutes, and 45 seconds becomes N10.3545E and the reverse bearing of 190 degrees, 35 minutes, and 45 seconds becomes S10.3545W.
Elevation	<p>Check mark this box to display elevation parameters on screen.</p>
Decimal Places	
	<p>Linear: Defines the number of decimal places to which values of distance, height, and area are displayed in input fields (default is 3 decimal places).</p> <p>Angular: Defines the number of decimal places to which angular measurements are displayed in input fields. When using Degrees, Minutes, and Seconds for angular measure, this must be set to four decimal places. Two decimal places would display only the Degree and Minute values.</p>
Print	<p>Prints unit settings.</p>
Load	<p>Loads a previous program configuration.</p> <p>Allows two or more users with different preferences to save and recall their settings.</p>
Save	<p>Saves the program configuration.</p>
OK	<p>Saves the settings you selected.</p>
Cancel	<p>Press to quit.</p>

Symbols

The *Symbols* tab establishes the ASCII characters used for specific symbols associated with annotating survey and engineering work (Figure 8-15 on page 8-18). These settings should only be modified if

MAGNET Office is being used in a country where different characters are used.



Figure 8-15. Program Settings – Symbols Tab

You can write some of these characters in a text string using the ASCII number if the symbol is not accessible from the keyboard. For example, to write the degree symbol in a text string, press and hold the **Alt** key and type 0176 (**Alt+0176**).

To enter a space in a field for such options as *Text Before* and *Text After* a label, use **Alt+0160**.

Coordinate System

To work in a predefined coordinate system, do the following:

1. Specify the coordinate system from the list of provided systems in the **Select projection** window (Figure 8-16 on page 8-19).
The parameters for the selected system are displayed in the **Selected parameters** window on the right.
2. Select the required **Projection ellipsoid**.

3. Select a **Geoid file** to use in conjunction with your defined settings if needed

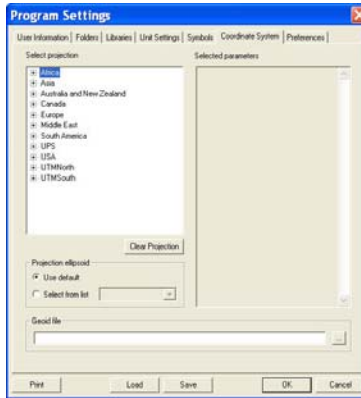


Figure 8-16. Program Settings – Coordinate System Tab

Preferences

Use the *Preferences* tab to set preferences that will define settings for all new projects. You can edit these settings at another time (Figure 8-17 on page 8-20).

Table 8-3 on page 8-20 describes the *Preferences* tab fields.

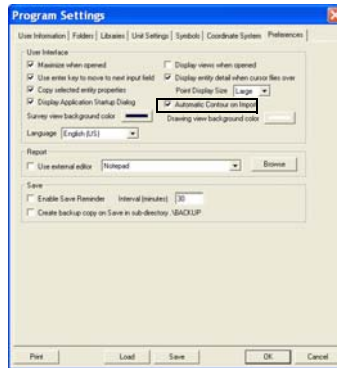


Figure 8-17. Program Settings – Preferences

Table 8-3. Preferences – Field Descriptions

Field Name	Field Description
User Interface	
Maximize when opened	Check mark this box to maximize the window of various views when a project is created or opened. If this check box is not marked, the views will be displayed in a normal window.
Use enter key to move to next input field	Check mark this box to input data by pressing Enter to move to the next data input field on the Input Bar. If this check box is not marked, press Tab to move to the next data input field and press the Enter key to finish and apply the data.
Copy selected entity properties	Check mark this box to copy the properties of one (selected) entity to apply them to new entities that are connected.
Display Application Startup Dialog	Check mark this box to display text upon startup of selected application.
Survey view background color	Select a background color from the drop-down entry box for the Survey View.
Language	Select the preferred language
Display views when opened	Check mark this box to open any existing view in the project when the project is opened. Each view will have a tab at the bottom of the screen. To access a view, click on the appropriate tab.

Table 8-3. Preferences – Field Descriptions (Continued)

Field Name	Field Description
Display entity detail when cursor flies over	Check mark his box to display the type and information about an entity when the pointer passes over it.
Point Display Size	Select Small or Big from the drop-down menu. This is the display size on screen only.
Automatic Contour on Import	<p>Check mark to automatically create contours from imported data that contains data with an elevation. The prerequisites for this option is to have a code library that matches those of the imported data. These codes should each be set to “contourable”. In most cases, the codes of the imported points will be unknown. Preferably, you should set the code named, “Default” to contourable, so that all imported data with an unknown code or without a code will be set to contourable. This allows the automatic contour setting to create both the contours and the DTM.</p> <p>NOTE: The contour settings will be based upon the Default contour settings defined in the Library menu (Contour Settings option),</p> <p>NOTICE: It is important to remember that this option only works using the import utility, however, opening a project will not automatically generate contours from the data.</p>
Drawing view background color	Select a background color from the drop-down entry box for the Drawing View.
Report	
Use external editor	Check mark this box to select an external editor for reports (optional). Select either <i>Notepad</i> , <i>Wordpad</i> , <i>Winword</i> , or <i>Excel</i> from the drop-down entry box. Press the Browse button for more options.
Save	
Enable Save Reminder	<p>Check mark this box to activate the MAGNET Office save reminder. The reminder will be displayed at the time set in the adjacent field (Interval (minutes)) after a routine is completed.</p> <p>If the box is not check marked, no reminder displays and you must remember to make regular saves.</p>

Table 8-3. Preferences – Field Descriptions (Continued)

Field Name	Field Description
Interval (minutes)	Enter a time in minutes between displays of the save reminder. You can choose to save or not save at that time.
Create backup copy on Save in sub-directory .BACKUP	Check mark this box to create a backup copy of the project when a Save command is used. The backup directory is made when you first save the project.
System Settings	
Smallest Triangle Side	Enter a parameter to check for duplicated points when creating a DTM.
Height Tolerance	Enter a parameter to compare points at the same position during the breakline process.
Roads	
Min/Max Super	Enter a minimum and a maximum superelevation for the curve.
Sewer Plot	Select either Drainage or Sewer for the plot setting.
Print	Prints unit settings.
Load	Loads a previous program configuration. Allows two or more users with different preferences to save and recall their settings.
Save	Saves the program configuration.
OK	Saves the settings you selected.
Cancel	Quits the routine.

Create Custom Colors

Use the *Create Custom Colors* dialog box to define the 16 basic colors (0-15) and additional non-standard colors using the custom colors palette.

To create a custom color, do the following:

1. Click **Settings ▶ Create Custom Colors**.

The *Create Custom Colors* dialog box appears (Figure 8-18).

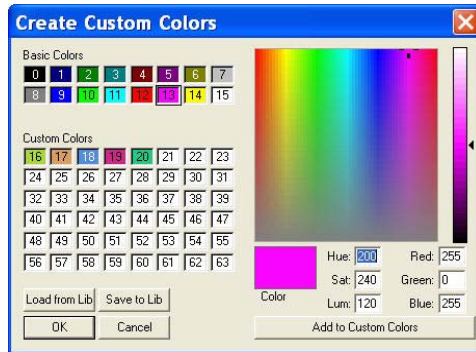



Figure 8-18. Create Custom Colors

2. Select a custom color number (24-63), and then a basic color (0-23).
The color options in this range display in the *Color* field on the right.
3. To customize a color, click in the shade bar or enter a numeric value in the fields.
4. Click on the **Add to Custom Colors** button to select and save a color to the selected number.
5. Click **Load from Lib** to import a color palette to the library.
6. Click **Save to Lib** to export a color palette from the library.
7. Click **OK** to save settings or **Cancel** to quit.

Display General Settings

Use the *Display* tab on the *General Settings* dialog box to control the display of entities and their properties. Entities on inactive layers will not be displayed in the Survey View.

To set display settings, do the following:

1. Click **Settings ▸ Display** or click the **Hot Button**  in the Status Bar.

The *General Settings* dialog box appears (Figure 8-19 on page 8-24).

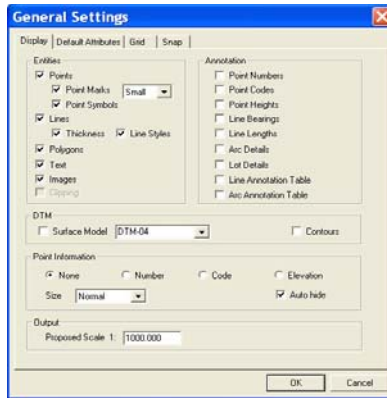


Figure 8-19. General Settings Dialog Box – Display Tab

2. Modify the properties as needed, and click **OK** when you are finished. Table 8-4 describes the *Display* fields.

Table 8-4. General Settings Display Tab Field Descriptions

Field Name	Field Description
Entities	
Points	Displays points. If this check box is not selected, then the points, point marks, and symbols are not displayed. <ul style="list-style-type: none"> • Point Marks: Displays point marks with the option to set their display on the screen to small or large marks. • Point Symbols: Displays point symbols.
Lines	Displays lines, arcs, circles, and strings, plus complex entities, such as curbs and alignments. If this check box is not marked, then the lines, line thickness and styles, arcs, circles, and strings do not display. <ul style="list-style-type: none"> • Thickness: Displays the line widths (thickness). • Line Styles: Displays the line styles
Polygons	Displays the polygons
Text	Displays text strings
Images	Displays images on screen.
Clipping	Only available when the project in Survey View is created from a drawing, using the Save As Project option in the Drawings View. This option ensures the data is clipped to the drawing area of the title block.

Table 8-4. General Settings Display Tab Field Descriptions (Continued)

Field Name	Field Description
Annotation	
Point Numbers	Displays the annotation for point numbers. Only points that have this option selected will be annotated with a point number.
Point Codes	Displays the annotation for point codes. Only points that have this option selected will be annotated with the data held in the Code field for a point.
Point Heights	Displays the annotation for point heights. Only points that have this option selected will be annotated.
Line Bearings	Displays the annotation for line bearings. Only lines that have this option selected will be annotated.
Line Lengths	Displays the annotation for line lengths. Only lines that have this option selected will be annotated.
Arc Details	Displays the annotation for arcs. Only arcs that have this option selected will be annotated.
Lot Details	Displays the annotation for lots. Only lots that have this option selected will be annotated.
Line Annotation Table	Displays the short line annotation table. Only lines that have the annotation and line annotation table options selected will be displayed in the table and numbered the same as that displayed by the relevant line.
Arc Annotation Table	Displays the arc annotation table. Only arcs that have the annotation and arc annotation table options selected will be displayed in the table and numbered the same as that displayed by the relevant line.
DTM (More than one DTM can display at the same time)	
Surface Model	Displays the DTM surface model (triangles) selected in the adjacent drop-down menu.
Contours	Displays the DTM contours selected in the Surface Model drop-down menu using the properties and contour intervals defined for the surface model. Set the format of the Contours display by clicking Surface ► DTM Settings .


Table 8-4. General Settings Display Tab Field Descriptions (Continued)

Field Name	Field Description
Point Information	<p>This option is for display only. The displayed text is non-scalable and cannot be printed or plotted. Only one label can be displayed at a time. Select either None, Number, Code, or Elevation.</p> <p>Auto hide: Indicates information is not displayed if the mapping is such that the data cannot be read.</p> <p>Select an option to display the attribute for all points on the active layers in the Survey View.</p>
Output	
Proposed Scale 1:	<p>The proposed scale is the intended output plot scale and enables the display of symbols, line types, and text as they will appear when printed or plotted at the scale defined in this field.</p>




To set the format for the annotation display, click **Settings ▶ Annotation Settings**.

Default Attributes

Use the *Default Attributes* tab on the **General Settings** (Hot Button ) dialog box (Figure 8-20 on page 8-27) to set various default settings that are applied to entities as they are created.

To define default attributes, do the following:

1. Click **Settings ▶ Default Attributes** or click the **Hot Button**  on the Status Bar.
2. Click on the *Default Attributes* tab.

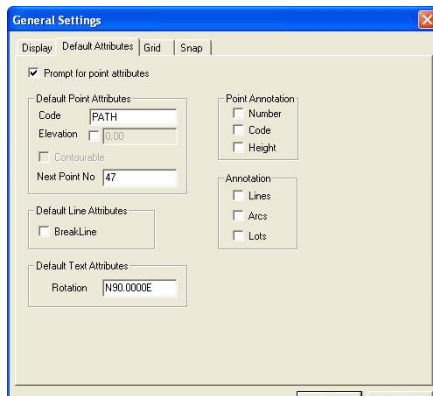


Figure 8-20. Default Attributes

3. Modify the fields as needed, and click **OK** when you are finished. Table 8-5 describes the Default Attributes fields.

Table 8-5. General Settings Default Attributes Tab Field Descriptions

Field Name	Field Description
Prompt for Point Attributes	Activates additional prompts on the input bars for entering more attribute information when creating entities.
Default Point Attributes	
Code	Type a code as the default for all new points.
Elevation	Activates the Elevation check box, and the value entered in the adjacent field is the default height allocated to all new points.
Contourable	Activates the Contourable option for all new points that have a default elevation applied. This check box is only available if the Elevation option is activated.
Next Point No	MAGNET Office automatically controls the numbering of points created in the Survey View. The default start point number in any new project is set to 1. This can be changed to a larger number before data is entered. The next point number is updated to the next available point number as data is created in the Survey View.

Table 8-5. General Settings Default Attributes Tab Field Descriptions (Continued)

Field Name	Field Description
Default Line Attributes	
Breakline	Activates the breakline option for all new lines.
Default Text Attributes	
Rotation	The settings in this field define the default rotation for all text as it is inserted. The default setting is dependent on the current setting of the Rotation By field on the Unit Settings tab of the Program Settings and Project Settings dialog boxes.
Point Annotation	
Number, Code, Elevation	Activates the setting for the individual Point Annotation option. All new points will have the annotation automatically set if one or more of the three check boxes are selected.
Annotation	
Lines, Arcs, Lots	Activates the setting for the individual Annotation option. All new lines, arcs, and lots will have the annotation automatically set if the appropriate box is check marked.

See “Point Creation Mode” on page 8-53 for details on the *Grid* and *Snap* tab on the *General Settings* dialog box.

Text Styles

The MAGNET Office library contains user-defined text styles. When a text style is used for text, annotation, or plotted in a project or drawing, the text style is embedded in the project so the style is maintained, regardless of the library settings, when the project is opened at a later date.

The *TEXT STYLES* dialog box (Figure 8-21 on page 8-29) lists the text styles used in the project and those embedded from the default annotation settings when the project was created.

To open the Text Style window, click **Settings ▶ Text Styles**. The *TEXT STYLE* dialog box appears (Figure 8-21).

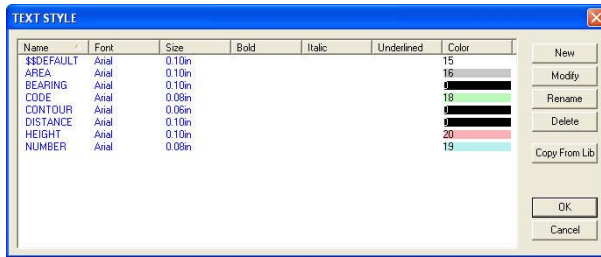


Figure 8-21. Text Style

This window lists the text styles already used in the current project and those activated in the *Annotation Settings* dialog box. Changes made to the Text Styles in this list only apply to the current job and do not modify the settings of the text styles in the library.

Each text style is defined with a font, size, font styles, and color.

Add Text Styles

To add a new text style, do the following

1. Click **New** on the *TEXT STYLE* dialog box.

A second *TEXT STYLE* dialog box appears (Figure 8-22).



Figure 8-22. Add Text Style

2. Enter a name for the text style and select a **Font**. A preview of the font type appears in the *Preview* box below (Figure 8-22).
3. Select a font **Size** from the drop-down entry box.

4. Select a **Color** from the drop-down entry box.
5. From the *Style* panel, select a style for the font type, either Bold, Italics, or Underline.
6. Click **OK** when you are finished.



The style is added to the list of text styles. It is not added to the library of text styles.

Modify Text Styles

To modify text styles, do the following:

1. Select a style name, and click **Modify**.

A new *TEXT STYLE* dialog box appears, displaying the current settings of the selected style (Figure 8-23).



Figure 8-23. Add Text Style

2. Make the necessary changes, and click **OK** when you are finished. MAGNET Office saves your changes to the text style list.

Copy a Text Style From the Library

To display a text style in the current project from the MAGNET Office library, do the following:

1. On the *TEXT STYLES* dialog box, click on the **Copy From Lib** button.

A list of text styles in the library display (Figure 8-24)



Figure 8-24. Text Style Library

2. Select the required text style from the list, and then click **OK**.
The selected text style is copied from the library into the current project and is displayed on the project list.

Justification Styles

The MAGNET Office library contains user-defined number justification styles. When numbers are entered as text, annotation, or plotted in a project or drawing, the format of the number and its decimal precision is defined by a justification style. The style is embedded in the project, so the style is maintained, regardless of the library settings, when the project is opened at a later date.

This option lists the justification styles used in the project and those embedded from the default annotation settings when the project is created.

To view the number justification styles, click **Settings** ► **Justification Styles**. The *Number Justification Style* dialog box appears (Figure 8-25 on page 8-32).

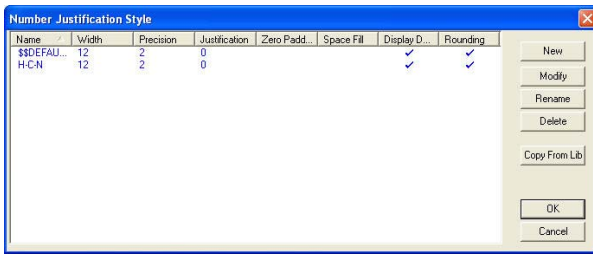


Figure 8-25. Number Justification Style

A list of justification styles already used in the current project, together with the default styles displays in the *Number Justification Style* dialog box. Changes made to the justification styles in this list only apply to the current project and do not modify the setting of the justification styles in the library.

The justification style controls the appearance of the numerical values used in such text items as height annotation, grid values, chainage, and levels in road drawings.

Each justification style is defined with a width, precision, justification position, and other options.

In This Section:

- “Add”
- “Modify” on page 8-34
- “Rename” on page 8-34
- “Delete” on page 8-34
- “Copy From Library” on page 8-35

Add

To create a new justification style for use in the current project, do the following:

1. Select a current style that has similar settings to the new style, and click **New**.

The *Number Style* dialog box appears (Figure 8-26 on page 8-33).

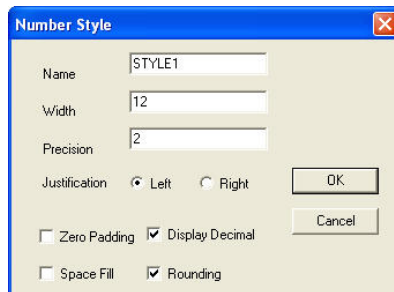


Figure 8-26. Number Style Dialog Box

2. Type a name in the appropriate field.
3. Type the number of digits used in the style in the **Width** field. This number includes the characters used for the decimal point and decimal figures.
4. Type the number of decimal places in the style in the **Precision** field.
5. Select either Left or Right for the justified position for the numerical text.
6. Select the following parameters by placing a check mark in the box next to them:
 - Zero Padding – check mark to display a prefixed number with zeroes to fill the character spaces in the width.
 - Space Fill – check mark to display a right-justified number with prefixed spaces to make up the number of characters in the width.
This option is ideal for positioning chainage and level values on profiles.
 - Display Decimal – check mark to display the decimal point. If the precision is set to 0, then the decimal point will be omitted if this check box is not selected.
 - Rounding – check mark to round a number to a precision setting.
7. Click **OK** to save the style to the list of number justification styles in the project. It is not added to the justification styles library.

Modify

To modify a justification style, do the following:

1. Select the style, and then click **Modify**.

The *Number Style* dialog box appears (Figure 8-27)



Figure 8-27. Number Style

2. Make the necessary changes, and click **OK** when you are finished.

MAGNET Office saves your changes to the justification style list.

Rename

To rename a justification file, do the following:

1. Highlight the file you want to rename.
2. Press **Rename**.
3. Press **OK** when done.

Delete

To delete a justification file, do the following:

1. Highlight the file you want to delete.
2. Press **Delete**.
3. Press **OK** to confirm deletion.

Copy From Library

To display a justification style in the current project from the MAGNET Office library:

1. On the *Number Justification Style* dialog box, click on the **Copy From Lib** button.

The list of justification styles in the library displays (Figure 8-28).

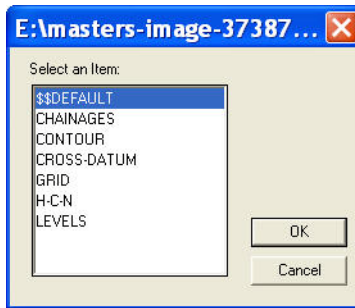


Figure 8-28. Justification Style Library

2. Select the required text style from the list, and then click **OK**.

The selected text style is copied from the library into the current project and is displayed on the project list.

Define Annotation Styles

Many of the entities in MAGNET Office have particular attributes that you can annotate on screen for display on a printed plan. A number of annotation styles are available to set appropriate annotation formats to suit international and national schemes.

When a new project is created, the default setting for each annotation setting and table is set in the project as the current style, and it will be used for any new annotations. The style applied to the annotation may be modified using the **Annotation Styles** command.

Libraries

Only one annotation setting or style is applied to an entity – points, lines, arcs, lots, and arrows/dimensions – at a time. These settings use other parameters from the annotation tables for bearing, distance, area, and angle.

Several annotation tables can be created or copied into the project from the library.

Annotation tables should be designed before the annotation settings are defined, as the settings styles refer to the tables.

To create and save annotation settings and tables in the MAGNET Office library, do the following:

1. Click **Settings** ▶ **Program Settings** ▶ **Libraries** (Figure 8-29).



Figure 8-29. Program Settings – Libraries Tab

2. Select the **Text Styles** library, and click **Edit**.

All of the annotation settings are held in this library and can be added to or modified using this option. Or, you can use the options in the **Library** menu.

Display Annotations

To display the annotation for the entities, the individual entity must have its annotation option selected, and it must also be selected in the **General Settings** (Hot Button) dialog box. The current annotation setting for the entity is applied when its annotation flag is turned on.

Distance Annotation Settings

Use the *Distance Annotation Settings* tab to apply a text style and rules for formatting distance text used for line, arc, and lot annotation (Figure 8-30). The Distance Annotation Settings also control the rounding of the distance text for each line to the number of decimal places required for a specified length of line.

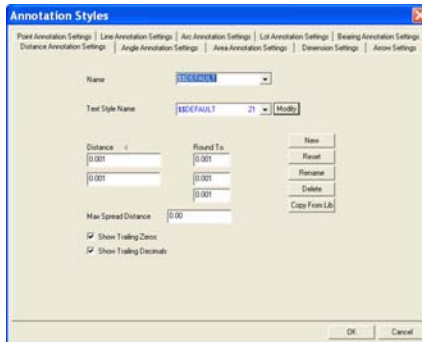


Figure 8-30. Distance Annotation Settings

Table 8-6 describes the Distance Annotation Settings fields.

Table 8-6. Distance Annotation Settings Field Descriptions

Field Name	Field Description
Name	Displays the name of the current table. The first table is called \$\$DEFAULT.
Text Style Name	Displays the text style used for the distance annotation of the lines. This can be modified by selecting an alternative text style from the drop-down menu.

Table 8-6. Distance Annotation Settings Field Descriptions (Continued)

Field Name	Field Description
Distance and Round To	<p>If the line has a distance less than the value entered in the Distance field (first line), then the annotation is rounded to the number of the value in the adjacent Round To field.</p> <p>If the line has a distance greater than the value entered in the first line, but less than the Distance field (second line), then the annotation is rounded to the number of the adjacent Round To field.</p> <p>For any greater distance, the annotation is rounded to the value in the last Round To field.</p> <p>A value of 0.010 will round distances to 2 decimals, while a value of 0.1 will round distances to 1 decimal.</p>
Max Spread Distance	Type a suitable value to spread the display of the annotation along the lines.
Show Trailing Zeros	Displays the trailing zeroes up to the number of decimals defined in the rounding settings.
Show Trailing Decimals	Displays the trailing decimal points for whole numbers.
New	Creates a new annotation table. You can accept the default table name or type a suitable name.
Reset	Resets annotation table entries to previous settings.
Rename	Renames an annotation table. Select the required table from the Name drop-down menu, and then select Rename to type a suitable name.
Delete	Deletes an annotation table. Select the required style from the Name drop-down menu, and then select Delete .
Copy from Lib	Copies the annotation settings from the library to the current project. Select the required style from the drop-down menu, and then click OK .

Angle Annotation Settings

Use the *Angle Annotation Settings* tab to apply a text style and rules for formatting angle text. These settings also control the rounding of the angles (Figure 8-31 on page 8-39).

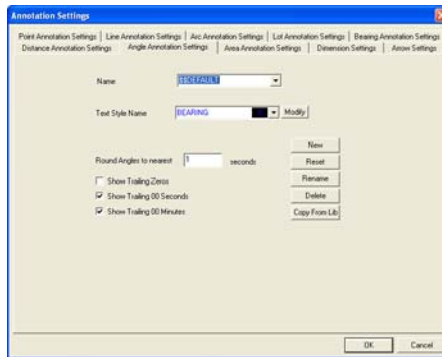


Figure 8-31. Angle Annotation Settings

Table 8-7 below describes the Angle Annotation Settings fields

Table 8-7. Angle Annotation Settings Field Descriptions

Field Name	Field Description
Name	Displays the name of the current table. The first table is called \$\$DEFAULT.
Text Style Name	Displays the text style used for the annotation of the angles. This can be modified by selecting an alternative text style from the drop-down menu.
Round Angles to Nearest	Type a value to round each angle to the number of seconds specified.
Show Trailing Zeros	Displays the trailing zeroes.
Show Trailing 00 Seconds	Indicates that the seconds value is displayed, even when the value is 00.
Show Trailing 00 Minutes	Indicates that the minutes value is displayed, even when the value is 00.
New	Creates a new annotation table. You can accept the default table name or type a suitable name.
Reset	Resets annotation table entries to previous settings.
Rename	Renames an annotation table. Select the required table from the Name drop-down menu, and then select Rename to type a suitable name.
Delete	Deletes an annotation table. Select the required style from the Name drop-down menu, and then select Delete .

Table 8-7. Angle Annotation Settings Field Descriptions (Continued)

Field Name	Field Description
Copy from Lib	Copies the annotation settings from the library to the current project. Select the required style from the drop-down menu, and then click OK .

Area Annotation Settings

Use the Area Annotation Settings tab to apply a text style and rules for formatting area text used for lot annotation. These settings also control the rounding of the areas and units used (Figure 8-32).

Table 8-8 describes the Area Annotation Settings field.

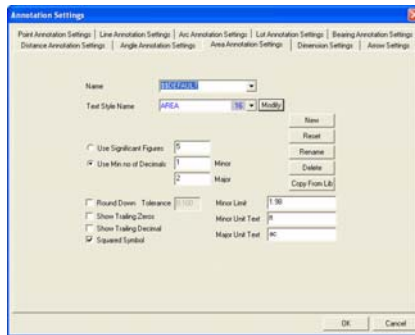


Figure 8-32. Area Annotation Settings

Table 8-8. Area Annotation Settings Field Descriptions

Field Name	Field Description
Name	Displays the name of the current table. The first table is called \$DEFAULT.
Text Style Name	Displays the text style used for the annotation of the area of lots. This can be modified by selecting an alternative text style from the drop-down menu.
Use Significant Figures	Displays the area annotation to the minimum number of significant figures entered in the adjacent field.
Use Min no of Decimals	Displays the area annotation to the minimum number of decimal places entered in the adjacent fields for both major and minor units of area.
Minor Limit	Type the largest value for the area that uses the minor unit before changing to the major unit.

Table 8-8. Area Annotation Settings Field Descriptions (Continued)

Field Name	Field Description
Minor Unit Text	Type the label used for the minor unit. For example: m will automatically display as m ² . Do not enter the 2.
Major Unit Text	Type the label used for the major unit.
Round Down	Indicates the area value is rounded down for the area annotation instead of using standard rounding rules. To use standard rounding rules, make sure the check box is cleared.
Show Trailing Zeroes	Displays the trailing zeroes up to the number of decimals defined above.
Show Trailing Decimal	Displays the trailing decimal points for whole numbers.
Squared Symbol	Displays the squared symbol in reports.
New	Creates a new annotation table. You can accept the default table name or type a suitable name.
Reset	Resets annotation table entries to previous settings.
Rename	Renames an annotation table. Select the required table from the Name drop-down menu, and then select Rename to type a suitable name.
Delete	Deletes an annotation table. Select the required style from the Name drop-down menu, and then select Delete .
Copy from Lib	Copies an annotation settings from the library to the current project. Select the required style from the drop-down menu, and then click OK .

Dimension/Arrow Settings

Use the *Dimension Settings* tab to set the style for creating dimensions. When you open the *Dimension Settings* tab (Figure 8-33), the current style for the project is displayed. In any

new project, the settings will be the same as the default style defined in the Library.

Arrow Settings Tab

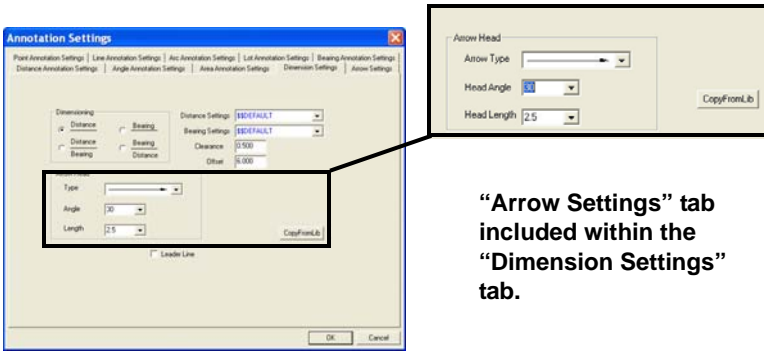


Figure 8-33. Dimension Settings

The settings in Figure 8-33 displays the annotation in Figure 8-34 when these entities are created using the options in the **Insert** menu.

Table 8-9 describes the *Dimensions and Arrow Settings* fields.

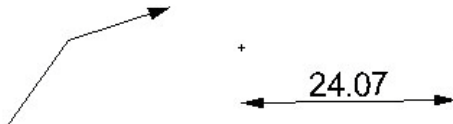


Figure 8-34. Arrows and Dimensions

The settings for arrows are applied when you create arrows using the **Insert** ▶ **Arrow** and **Insert** ▶ **Dimension** options.

Table 8-9. Dimensions and Arrow Settings Field Descriptions

Field Name	Field Description
Dimensioning	Select a format for the bearing and/or distance annotation. Options with a line indicate how the annotation is displayed relative to the arrow.
Distance Settings	Select a setting to allocate a text style and apply rounding to the dimension distance.
Bearing Settings	Select a setting to allocate a text style and apply the rounding to the dimension bearing.

Table 8-9. Dimensions and Arrow Settings Field Descriptions (Continued)

Field Name	Field Description
Clearance	Type the clearance distance between the line and the annotation measured in feet. For example: A clear distance of .2 at a 1" = 50' scale results in a distance of 1.
Offset	Type the offset distance between the line entity and the dimension line measured in feet. For example: An offset distance of .2 at a 1" = 50' scale results in a distance of 1.
Arrow Type	Select the arrow type from the drop-down menu.
Arrow Angle	Select the angle of the arrow head from the drop-down list or type the value.
Arrow Length	Select the length of the arrowhead in millimeters from the drop-down menu or type the value.
Leader Line	Adds extension lines to the dimensions.
Copy From Lib	Copies an annotation settings from the library to the current project. Select the required style from the drop-down menu, and then click OK .

Point Annotation Settings

Use the *Point Annotation Settings* tab to set the style for point annotations. When you open the *Point Annotation Settings* tab, the current style for the project is displayed. In any new project, the settings will be the same as the default style defined in the Library.

The point annotation is displayed for points that have any of their three point annotation flags selected, and the point annotation options are active in the *General Settings* (Hot Button) dialog box (Figure 8-35 on page 8-44).

Table 8-10 on page 8-44 describes the Point Annotation Settings fields.

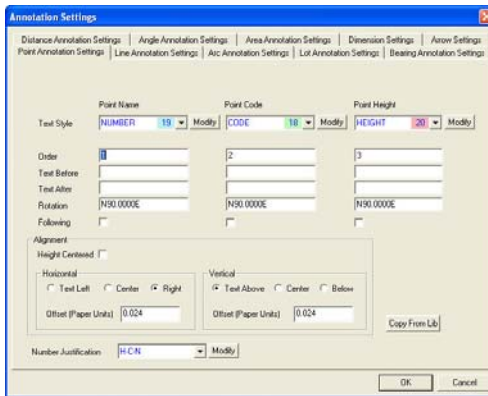


Figure 8-35. Point Annotation Settings

Table 8-10. Point Annotation Settings Field Descriptions

Field Name	Field Description
Text Style	Select the text style for the name, code, and height. These can be modified by selecting an alternative text style from the drop-down menu.
Order	Specify the order for positioning the name, code, and height. The attributes are positioned in order of 1, 2, 3, with 1 being the first attribute. If one of the 3 annotation flags for a point is not selected, then it will not be displayed in on the screen and only positions 1 and 2 will be used.
Text Before	Type any text for display before the name, code, or height, such as Ht= or ID=.
Text After	Type any text for display after the name, code, or height, such as m for meters after each height.
Rotation	Type the orientation of the annotation for the name, code, and height. By default this is 90.0000 (horizontal) for a rotation set to Bearing on the Unit Settings tab of the Program Settings dialog box. Modify the bearing to set the annotation at an angle.
Following	Indicates the annotation of the attribute set to Order=2 is placed to the right of the first attribute.
Height Centered	Indicates the height annotation is centered around the point entity.

Table 8-10. Point Annotation Settings Field Descriptions (Continued)

Field Name	Field Description
Text Left, Center, or Right	Select a position for the annotation of the first attribute in the order regarding the position of the point entity. For example: Text Right places the annotation to the right of the point.
Text Above, Center, or Below	Select a position for the annotation of the first attribute in the order regarding the position of the point entity. For example: Text Above places the annotation above the point.
Show Trailing Decimal	Displays the trailing decimal points for whole numbers.
Number Justification	Select a number justification for heights to control the number of decimal places used and the format of the height annotation.
Copy from Lib	Copies the annotation settings from the library to the current project. Select the required style from the drop-down menu, and then click OK .

Line Annotation Settings

Use the *Line Annotation Settings* tab to set the style for line annotations. When you open the *Line Annotation Settings* tab, the current style for the project will display. In any new project, the settings will be the same as the default style defined in the Library.

The line annotation is displayed for lines that have any of their line annotation flags selected, and the line annotation options are active in the *General Settings* (Hot Button) dialog box (Figure 8-36 on page 8-46).

Table 8-11 on page 8-46 describes the Line Annotation Settings fields.

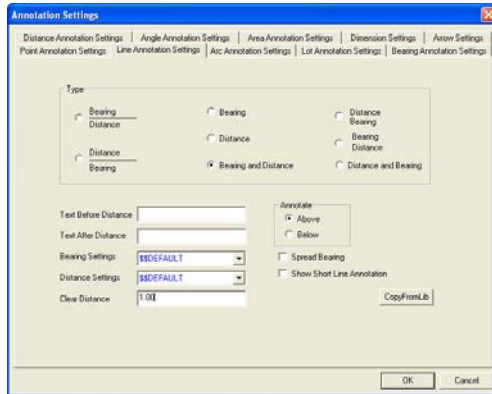


Figure 8-36. Line Annotation Settings

The settings in Figure 8-36 displays the annotation in Figure 8-37. The line entity has the annotation flag turned on.

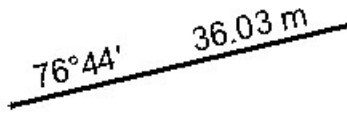


Figure 8-37. Line Annotation

Table 8-11. Line Annotation Settings Field Descriptions

Field Name	Field Description
Type	Select the format for the bearing and/or distance annotation. A line in the type indicates the annotation is displayed relative to the line. Options with no drawn line will be set above or below the line entity.
Text Before Distance	Type any text for display before the distance.
Text After Distance	Type any text for display after the distance, such as m for meters.
Bearing Settings	Select the Bearing Settings used to define the text style used and the rounding parameters for the bearing. These can be modified by selecting an alternative setting from the drop-down list.
Distance Settings	Select the Distance Settings used to define the text style used and the rounding parameters for the distance. These can be modified by selecting an alternative setting from the drop-down menu.

Table 8-11. Line Annotation Settings Field Descriptions (Continued)

Field Name	Field Description
Clear Distance	Type the clearance distance between the line and the annotation, measured in feet. For example: A clear distance of .2 at a 1" = 50' scale results in a distance of 1.
Annotate	Choose whether the annotation is displayed above or below the appropriate line. This setting is only available where the type has no drawn line.
Spread Bearing	Indicates the bearings will be spread along the lines, depending on the Bearing Settings.
Show Short Line Annotation	Indicates the lines will be annotated with bearing and distance even if the line is short and the annotation does not fit easily along the line. If not selected, then the line will only be annotated in the short line settings.
Copy from Lib	Copies the annotation settings from the library to the current project. Select the required style from the drop-down menu, and then click OK .

Arc Annotation Settings

Use the *Arc Annotation Settings* tab to set the style for arc annotations. When you open the *Arc Annotation Settings* tab, the current style for the project displays. In any new project, the settings will be the same as the default style defined in the Library.

The annotation is displayed for arcs that have any of their arc annotation flags selected, and the arc annotation options are active in the *General Settings* (Hot Button) dialog box (Figure 8-38 on page 8-48).

Table 8-12 on page 8-48 describes the Arc Annotation Settings fields.

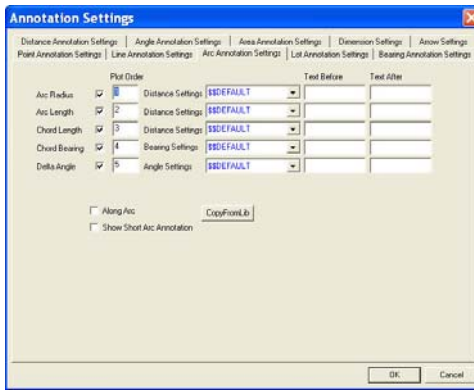


Figure 8-38. Arc Annotation Settings

The settings in Figure 8-38 displays the annotation in Figure 8-39. The arc entity has the annotation flag turned on.

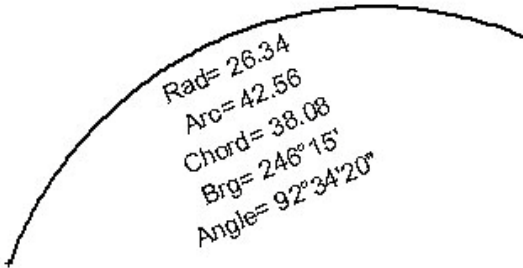


Figure 8-39. Arc Annotation

Table 8-12. Arc Annotation Settings Field Descriptions

Field Name	Field Description
Arc Radius	Displays the arc radius and type in order for plotting.
Distance Settings	Allocates a text style and applies rounding to the arc radius
Text Before	Type any text required before the radius annotation.
Text After	Type any text required after the arc length annotation.
Arc Length	Displays the arc length and type in order for plotting.
Distance Settings	These settings allocate a text style and apply rounding to the arc length.
Text Before	Type any text required before the arc length annotation.
Text After	Type any text required after the chord length annotation.

Table 8-12. Arc Annotation Settings Field Descriptions (Continued)

Field Name	Field Description
Chord Bearing	Displays the chord bearing and type in order for plotting.
Bearing Settings	These settings allocate a text style and apply rounding to the chord bearing.
Text Before	Type any text required before the chord bearing annotation.
Text After	Type any text required after the chord bearing annotation.
Angle Settings	These settings allocate a text style and format, and apply rounding to the arc angle.
Text Before	Type any text required before the arc angle annotation.
Text After	Type any text required after the arc angle annotation.
Delta Angle Set	This allocates a text style and applies rounding to the angle.
Text Before	Type any text required before the arc delta angle annotation.
Text After	Type any text required after the arc delta angle annotation.
Along Arc	Indicates the annotation is spread along the arc with the items positioned as one following the other.
Show Short Arc Annotation	Indicates the arcs will be annotated with the specified details even if the arc is short and the annotation does not fit easily along the arc. If not selected, the arc will only be annotated in the Short Line Table.
Copy From Lib	Copies the annotation settings from the library to the current project. Select the required style from the drop-down menu, and then click OK .

Lot Annotation Settings

Use the *Lot Annotation Settings* this tab to set the style for lot annotations. When you open the *Lot Annotation Settings* tab, the current style for the project is displayed. In any new project, the settings will be the same as the default style defined in the Library.

The annotation is displayed for lots that have any of their lots annotation flags selected, and the lots annotation options are active in the *General Settings* (Hot Button) dialog box (Figure 8-40 on page 8-50).

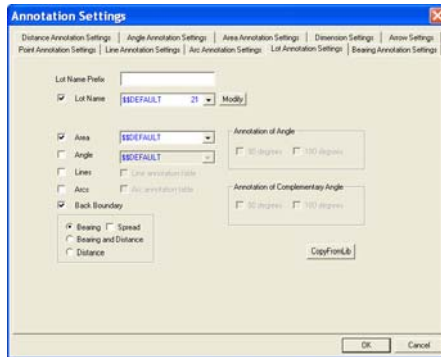


Figure 8-40. Lot Annotation Settings

Table 8-13 describe the Lot Annotation Settings fields.

In Figure 8-41, the lot entity has the annotation flag turned on.



Figure 8-41. Lot Annotation

Table 8-13. Lot Annotation Settings Field Descriptions

Field Name	Field Description
Lot Name Prefix	Type any text before the lot number
Lot Name	Displays the lot name (usually a number). Select a text style from the drop-down menu.
Area	Displays the lot areas. Select an area annotation from the drop-down menu.
Angle	Displays lot angles. Select an angle annotation from the drop-down menu.
Lines	Displays the lines that define the lot boundaries.
Arcs	Displays the arcs that define the lot boundaries.

Table 8-13. Lot Annotation Settings Field Descriptions (Continued)

Field Name	Field Description
Back Boundary	Select the appropriate format for annotating the lot back boundaries.
Annotation of Angle	If angles are included in the annotation, then select an appropriate degree.
Annotation of Complementary Angle	If complementary angles are included in the annotation, then select an appropriate angle.
Copy From Lib	Copies the annotation settings from the library to the current project. Select the required style from the drop-down menu, and then click OK .

Bearing Annotation Settings

Use the *Annotation Styles* dialog box (Figure 8-42) to apply text styles and rules to format bearing text used for line, arc, and lot annotation. These settings also control the rounding of the bearing text for each line to the required number of seconds for a specified length of line. See Table 8-14 on page 8-52 for Bearing Annotation field descriptions.

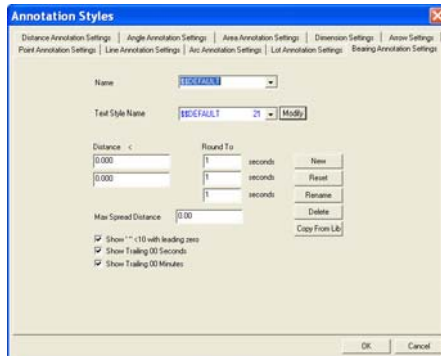


Figure 8-42. Bearing Annotation Settings





Table 8-14.

Field Name	Field Description
Name	Displays the name of the current table. The first table is called \$\$DEFAULT.
Text Style Name	Displays the text style used for the bearing annotation of the lines. This can be modified by selecting an alternative text style from the drop-down menu.
Distance and Round To	<p>If the line has a distance less than the value entered in the Distance field (first line), then the annotation is rounded to the number of the value in the adjacent Round To field.</p> <p>If the line has a distance greater than the value entered in the first line, but less than the Distance field (second line), then the annotation is rounded to the number in the adjacent Round To field.</p> <p>For any greater distance, the annotation is rounded to the value in the last Round To field.</p> <p>A value of 0.010 will round distances to 2 decimals, while a value of 0.1 will round distances to 1 decimal.</p>
Max Spread Distance	Type a suitable value to spread the display of the bearing annotation along the lines.
Show “<10 With Leading Zero	Indicates that minutes and seconds values less than 10 display with a zero in front (eg. 08).
Show Trailing 00 Seconds	Indicates that the seconds value is displayed, even when the value is 00.
Show Trailing 00 Minutes	Indicates that the minutes value is displayed, even when the value is 00.
New	Creates a new annotation table. You can accept the default table name or type a suitable name.
Reset	Resets annotation table entries to previous settings.
Rename	Renames an annotation table. Select the required table from the Name drop-down menu, and then select Rename to type a suitable name.
Delete	Deletes an annotation table. Select the required style from the Name drop-down menu, and then select Delete .
Copy from Lib	Copies the annotation settings from the library to the current project. Select the required style from the drop-down menu, and then click OK .

Point Creation Mode

Absolute positioning defines the point position by coordinate values, usually East and North of an origin, while relative positioning fixes the position of one point relative to another, usually as a radiation, by bearing or angle and distance. MAGNET Office accommodates both the coordinate and radiation modes, and also latitude and longitude modes, and can be easily switched from one to the other.

There are four input or point creation modes available:

- Point Coordinates  – See “Point Coordinates Mode” on page 8-54.
- Point Latitude and Longitude  – See “Point Latitude and Longitude Mode” on page 8-55.
- Point Radiation  – See “Point Radiation Mode” on page 8-55.
- Points Mode  – See “Create Points Mode” on page 8-57.

Click **Settings** ► **Create Mode** (Figure 8-43) to activate the input or creation modes or click the desired mode on the **Input Modes** toolbar. These modes are mutually exclusive – when one mode is activated the other is deactivated.

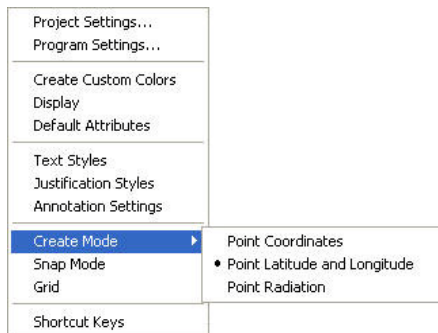


Figure 8-43. Create Mode

Many options automatically set the **Input Modes** toolbar to a suitable setting. For example, the **Add Line** icon switches to **Point Radiation** mode, assuming that the point at the end of the line will be created by a bearing and distance from the first point. The **Add Point** icon

switches to **Point Coordinate** mode to create points at Easting and Northing coordinate positions. This can be changed into **Radiation** mode once the start point has been selected, allowing points to be created at a bearing and distance from the initial point.





The Create Mode may be changed once a routine has been started – just click the icon for the required mode.

In This Section:

- “Point Coordinates Mode” on page 8-54
- “Point Latitude and Longitude Mode” on page 8-55
- “Point Radiation Mode” on page 8-55
- “Create Points Mode” on page 8-57

Point Coordinates Mode

To activate this mode, click **Settings ▶ Create Mode ▶ Point Coordinates**. You can also click the *Add a Point*  icon on the Add Entity toolbar, and then click the *Set Point Coordinate*  icon on the Input Modes toolbar.

When the **Point Coordinates Mode** is set (Figure 8-44 on page 8-55), then all operations are based on a methodology using the Cartesian plane or plane grid. Points are identified by their position as a distance in an East or X direction and a distance in a North or Y direction from an origin. In this manner, each point is located independently of every other point.

When creating a point in this mode, the prompt line requests you to define the point position by entering the Easting and Northing coordinates or by clicking on the screen.

Number	502	East	425126.200	<input checked="" type="checkbox"/>	North	804569.35	<input checked="" type="checkbox"/>
--------	-----	------	------------	-------------------------------------	-------	-----------	-------------------------------------

When data is being moved to a new position, the movement is defined by X and Y offsets.

X Offset	0.000	Y Offset	0.000	Rotation	0.0000	Scale	1.000000
----------	-------	----------	-------	----------	--------	-------	----------

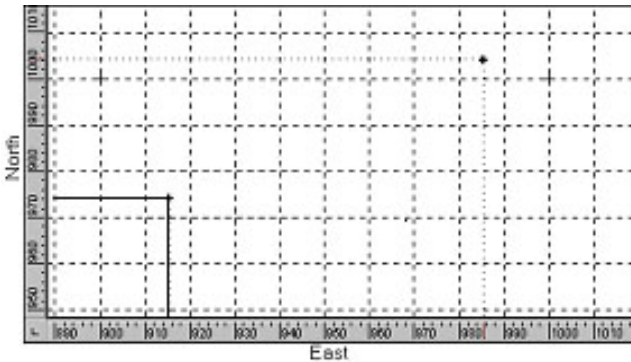




Figure 8-44. Defining Position by Coordinates

Point Latitude and Longitude Mode



Use the Point Latitude and Longitude Mode command to activate points located on the drawing screen by their latitude and longitude.

To activate this mode, click **Settings** ▶ **Create Mode** ▶ **Point Latitude and Longitude**. You can also click the *Add a Point*  icon on the **Add Entity** toolbar, and then click the *Set Point Latitude and Longitude*  icon on the **Input Modes** toolbar. The input bar appears.



A coordinate system must be specified for this command to work.

Point Radiation Mode

To activate this mode, click **Settings** ▶ **Create Mode** ▶ **Point Radiation**. You can also click the *Add a Point*  icon, and then the *Set Point Radiation*  icon on the Input Modes toolbar.

When the Point Radiation Mode is set then all operations are based on a methodology using a polar grid. Points are identified by their position as an angle and a distance from another point. In MAGNET Office, the angle is generally measured in a clockwise direction though there are instances where a counter clockwise direction can be

chosen. The zero position for measurement of an angle is determined by the option being used.

A specific form of the angle is called a bearing, which is measured in a clockwise direction with the zero point set at North Azimuth. Click **Settings ▶ Program Settings ▶ Unit Settings** to set the zero point to South Azimuth. In the *Unit Settings* dialog box change the Base Horizontal Angle.

When creating a point in this mode, the start point is selected and the prompt line then requests you to define the following point positions by entering the bearing and distance from the start point to the new point or by clicking on the screen (Figure 8-45).

Number	50	Bearing	110.0000	<input type="checkbox"/>	Distance	16.500	<input type="checkbox"/>	
--------	----	---------	----------	--------------------------	----------	--------	--------------------------	---

When data is being moved to a new position, the movement is also defined by bearing and distance.

Bearing	0.0000	Distance	0.000	Rotation	0.0000	Scale	1.000000
---------	--------	----------	-------	----------	--------	-------	----------

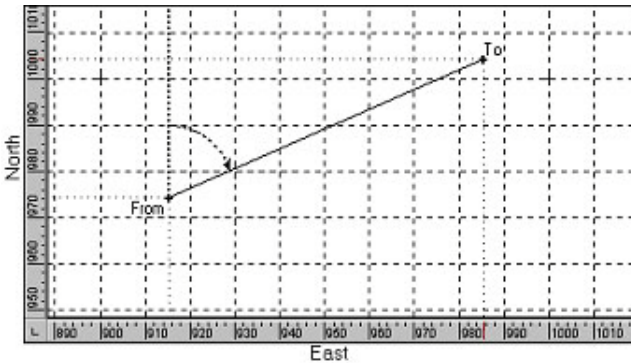


Figure 8-45. Defining Position Using Vectors

Create Points Mode


When Create Points Mode is switched **On**, points will be created as usual, however, if no points are required (only linework) switch **Off** create points mode.



Click **Insert ▶ Polyline** to automatically deactivate the “Create Points” mode, to draw linework without points. This is used more for drafting purposes. (Alternatively, **Insert ▶ String** will create Points).

Apply the Snap Mode

When a point is created, it is usually required to be in a specific location. For example: you might need to place a point exactly on a line. Depending on the viewing scale, the point may appear to be on the line at small scales but at a larger scale, this may not be the case. The purpose of the Snap options is to force a point exactly to the required location.

When using the snap modes (Figure 8-46 on page 8-58), it may be necessary to set or adjust the snap tolerance, which determines the distance the cursor must be away from the point before the point is no longer recognized. To adjust the Snap Mode settings, click **Settings ▶ Snap Mode**. You can also click the **Hot Button**  and select the *Snap* tab.

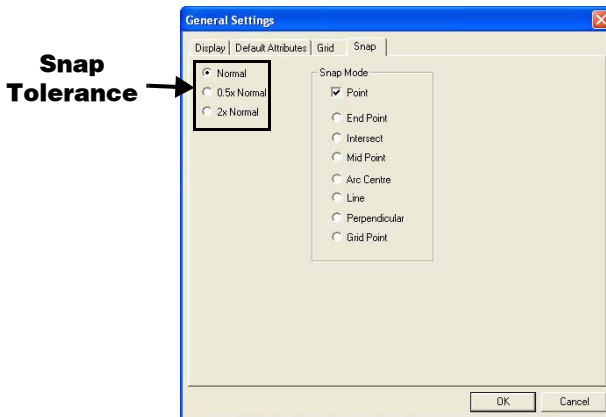


Figure 8-46. Snap Mode Settings

The snap tolerance is the proximity of the cursor to a point such that the point can be selected. It is measured by the size of the cross hairs of the cursor. The snap will be to the point closest to the center of the cursor or to the entity closest to the center. At the normal setting, the cursor is the standard size. At the 0.5x setting, the cursor is half of the standard size. At the 2x setting, the cursor is twice the standard size.

Each mode is activated by selecting it on the *Snap* tab and then clicking **OK** or by clicking its icon on the Snap Modes toolbar (Figure 8-47).



Figure 8-47. Snap Modes Toolbar

All modes, except for the Point Snap mode, are mutually exclusive. When one mode is activated the other is deactivated. Point Snap mode remains on when at least one other snap mode is selected.

Point Snap Mode

When a point is created, the cursor will snap to the closest point within the snap tolerance. If data is being selected, the cursor will snap to and highlight the nearest entity within the snap tolerance. If the wrong entity is selected, right-click, then select **Next Selection** if it is available in the shortcut menu.

End Point Snap

When a point is created, the cursor will snap to the end point of the line or arc within the snap tolerance. For example: use Add Line to create a new line that is to snap directly to the end point of an existing line. Click the existing line, and the new line will be attached to the nearest end point of the selected line. This is useful when lines must attach to specific points to ensure continuity of a string, lot, or alignment definition.

Intersect Snap

When a point is created, the cursor will snap to the intersection of two crossing features within the snap tolerance. For example: use Add Line to create a new line that is to snap directly to the position where two entities cross. Click on the intersection point of the two entities, and the new line will be located exactly at the intersection point.

Mid Point Snap

When a point is created, the cursor will snap to the mid point of the line or arc selected within the snap tolerance. For example: use Add Line to create a new line that is to snap directly to the mid point of an existing line. Click on the existing line, and the new line will be located exactly at the mid point of the selected line.

Arc Center Snap

When a point is created, the cursor will snap to the center point of the arc selected within the snap tolerance. For example: use Add Point to create a center point for a 3-point arc. Click on the existing arc and a point is created at the arc center point.

Line Snap

When a point is created, the cursor snaps to the nearest line or arc selected within the snap tolerance. For example: use Add Line to create a new line that will finish with its end point exactly on an existing line. With Line Snap mode, click on the existing line and the

new line will have its end point exactly on the selected line. This is useful when you know the bearing of a line but its distance is based on the position of another line.

Perpendicular Snap

When a point is created, the cursor will snap perpendicular to the line or arc selected within the snap tolerance. For example: use Add Line to create a new line that is to snap perpendicular to an existing line. With Perpendicular Snap, click on the existing line with the mouse and the new line will be attached perpendicular to the selected line.

Grid Point Snap

When a point is created, the cursor will snap to the grid intersection point within the snap tolerance. The grid is defined and activated using the **Grid** tab in the **General Settings** (Hot Button) dialog box. For example: use Add Point to create a number of points exactly at the intersection points of the grid. With the grid spacing defined, the grid displayed on the screen, and the grid snap mode set, click on the grid intersection points, which will create points with the exact coordinates of the specified grid.

Shortcut Keys

The Shortcut Keys command enables you to assign MAGNET Office functions to keys on the keyboard. Some shortcuts are already assigned.

To customize shortcut keys:

1. Click **Settings** ▶ **Shortcut Keys**.
The *Shortcut Keys* dialog box appears (Figure 8-48 on page 8-61).

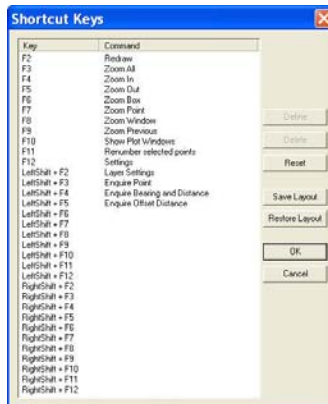


Figure 8-48. Shortcut Keys

2. Select a shortcut in the **Key** column, then click **Define**. The **Command** drop-down menu appears (Figure 8-49).

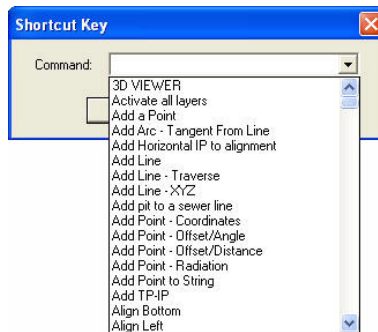


Figure 8-49. Command Drop-Down Menu

3. Select an option from the **Command** drop-down menu, and then click **OK** to save.
4. On the **Shortcut Key** dialog box, click **OK** to activate the shortcut keys you selected.

Notes:

Survey View: Insert Menu

This section describes the different Survey View menu commands on the Insert menu located on the main menu bar (Figure 9-1).

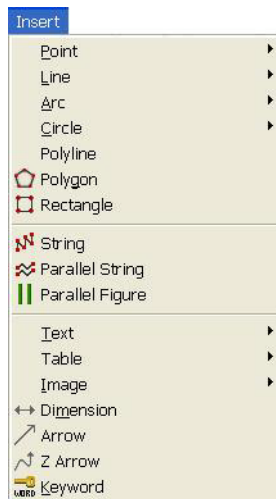


Figure 9-1. Survey View – Insert Menu

In This Section:

- “Create a Point” on page 9-2
- “Lines” on page 9-8
- “Define an Arc” on page 9-19
- “Define a Circle” on page 9-31
- “Polylines” on page 9-33
- “Polygons” on page 9-33
- “Rectangle” on page 9-36

- “Strings” on page 9-37
- “Parallel String” on page 9-40
- “Parallel Figure” on page 9-40
- “Define Text” on page 9-43
- “Custom Tables” on page 9-53
- “Image” on page 9-60
- “Dimension” on page 9-64
- “Insert Arrow” on page 9-65
- “Z Arrow” on page 9-66
- “Keywords” on page 9-67

Create a Point

The point is the main entity in MAGNET Office. All other entities are constructed using a point or points as the foundation. A point represents a unique position and may be a survey point, a designed point, a construction point to aid computation, or a feature point (Figure 9-2 on page 9-3).

A point has coordinates (Easting, Northing, or X, Y value) to locate the point in terms of the datum or coordinate system of the dataset. It may also have an elevation (height or z value). Each point has a unique alphanumeric number to identify it in the MAGNET Office database.

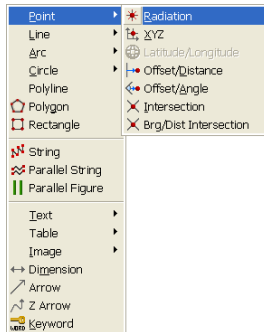


Figure 9-2. Point Submenu

In This Section:

- “Radiation” on page 9-3
- “XYZ” on page 9-4
- “Offset/Angle” on page 9-5
- “Intersection” on page 9-6
- “Bearing/Distance Intersection” on page 9-8

Radiation

Use the Radiation command to create a point or several points by entering a bearing and distance from a selected existing point (Figure 9-3).

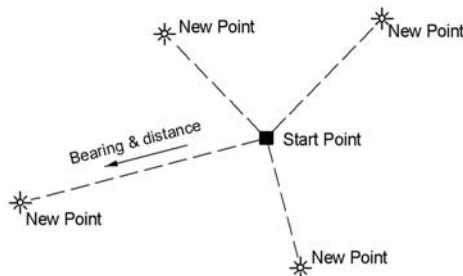


Figure 9-3. New Points Created by Radiation

To use this option, see “Point Radiation Mode” on page 8-55.

XYZ

Use the XYZ command to create a point or points by entering Easting and Northing coordinates.

To use the XYZ command, do the following:

1. Click **Insert ▶ Point ▶ XYZ**. When this option is used, the Point Coordinate mode is set and the input bar displays (Figure 9-4).

Number	502	East	425126.200	<input checked="" type="checkbox"/>	North	804569.35	<input checked="" type="checkbox"/>
--------	-----	------	------------	-------------------------------------	-------	-----------	-------------------------------------

Figure 9-4. Input Bar

2. Click the start point or type the coordinates into the input bar and press **Enter** to create the point.
3. Repeat step 2 to create additional points as needed. For more information about entering numerous points, see “Manual Data Entry” on page 12-5.
4. Click on the start point or type the latitude and longitude coordinates into the input bar and press **Enter** to create the point.

Repeat this process as needed to create additional points.

For more information about entering numerous points, see “Manual Data Entry” on page 12-5.

Offset/Distance

Use the Offset/Distance command to create a point at a distance along a reference line, arc, string, or alignment offset by a perpendicular distance. The reference entity may be an existing line, arc, string or alignment, or a line defined by any two points as start and end points (Figure 9-5).

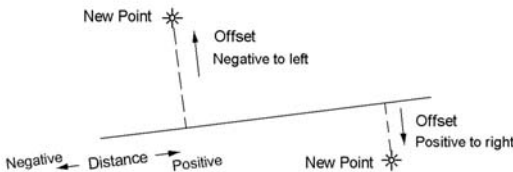


Figure 9-5. New Points Created by Offset and Distance

To use the Offset/Distance command, do the following:

1. Click **Insert ▶ Point ▶ Offset/Distance**.
2. Select a point, line, arc, string, or alignment from which the point will be offset.
3. Locate the point position by entering the perpendicular offset from the reference entity and the distance along the reference entity in the input bar on the lower left of the window (Figure 9-6).

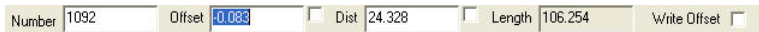


Figure 9-6. Input Bar

4. To end this option, press **Esc**, or select **Cancel** from the right-click shortcut menu.



You can switch the Input mode from Coordinate mode to Radiation mode and continue to create points at a distance offset from the same reference line and start point.

Offset/Angle

Use the Offset/Angle command to create a point as a radiation from a start point. The angle of the radiation is offset from a reference line, arc, string, or alignment. The distance is measured from the start point. The reference line may be an existing line, arc, string or alignment, or a line defined by any two points as start and end points (Figure 9-7).

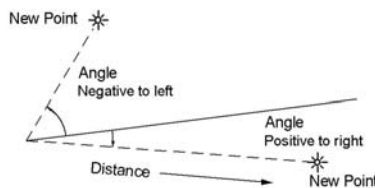


Figure 9-7. New Points Created by Angle and Distance

To use the Offset/Angle command, do the following:

1. Click **Insert ▶ Point ▶ Offset/Angle**.

2. Select a point, line, arc, string, or alignment from which the point will be offset.
3. Locate the point position as a radiation by entering the angle offset from the reference entity and the distance from the start point in the input bar on the lower left of the display area.
4. To end this option, press **Esc**, or select **Cancel** from the right-click shortcut menu.



You can switch the Input mode from Radiation mode to Coordinate mode and continue to create points at an angle offset from the same reference line and start point.

Intersection

Use the Intersection command to create a point or points at the intersection of lines, arcs, and circles with the provision to be offset from the chosen entity. A line may also be defined by selecting a starting point and an ending point (Figure 9-8 and Figure 9-9).

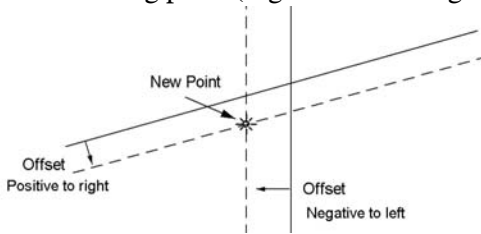


Figure 9-8. Point Created at Intersection of Two Offset Lines

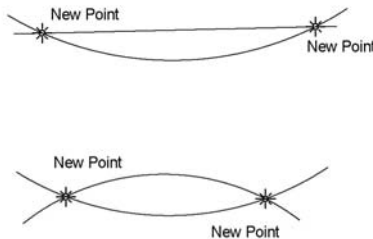


Figure 9-9. Two Possible Solutions: Two Arcs or an Arc and a Line

To use the Intersection command, do the following:

1. Click **Insert ▶ Point ▶ Intersection**.
2. Select a point as the start point of the first line or select the first line, circle, or arc.
3. If you selected a start point, then a second point is required to define the line. Do one of the following:
 - Click to select the second point.
 - Enter the point number into the Input Bar.
 - Enter a bearing to the second point into the Input Bar (Figure 9-10).

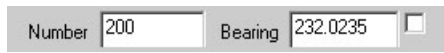


Figure 9-10. Input Bar

4. Enter the offset or accept the default setting of zero, and press **Enter**. The position of the first line, circle, or arc is displayed at the appropriate offset.
5. Select a point as the start point of the second line or select the second line, circle, or arc.
6. If a start point is selected, then a second point is required to define the line (see Step 3 on page 9-7).
7. Enter the offset or accept the default setting of zero, and press **Enter** to create the point(s). A point is created at the intersection of the selected entities.

When two solutions are possible, the prompt line provides a choice to save the current point.

8. Save the required point position by pressing **Enter** and discard the unwanted point using the **Esc** key or right-mouse shortcut menu.

3D Line If the entities selected define 3D lines (lines connecting points with elevations), then the **Intersection Point Height** dialog box appears (Figure 9-11 on page 9-8).

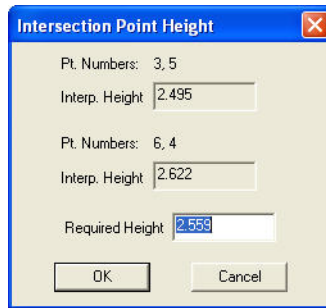


Figure 9-11. Point Height May Be Applied

The required height is the average of the Interpolated Heights. You can accept or edit this height, and then click **OK**.

Bearing/Distance Intersection

To create a point based upon a specified bearing or distance from two points:

1. Click **Insert ▶ Point ▶ Brg/Dist Intersection**.

The input bar and prompt line appear.

2. Click on a point or line.
3. If you selected a line, the existing line bearing is used for the intersection bearing.

If you selected a point, then enter the bearing and distance into the input bar. The prompt line requests an offset distance. Enter the offset or press **Enter** to continue.

4. Select the second point, and repeat Step 3 above.

Lines

A line represents the joining of two points. Its position is totally dependent on the points it joins. The line will be a 3D line if both points that define the line have elevation values. A line may also be defined as a breakline for use in a surface model.

In This Section:

- “Traverse Line” on page 9-9
- “XYZ Line” on page 9-10
- “Join Points” on page 9-11
- “Join Two Points” on page 9-12
- “Perpendicular Line” on page 9-12
- “Parallel Line” on page 9-13
- “Tangent Line” on page 9-15
- “External Tangent Line” on page 9-16
- “Splay Corner Command” on page 9-17
- “Fit Line” on page 9-18
- “Rectangle as Lines” on page 9-18

Traverse Line Use the Traverse command to create a traverse line by entering bearing and distances from a start point to create new points. The created points will be joined by lines along the length of the individual traverse line (Figure 9-12).

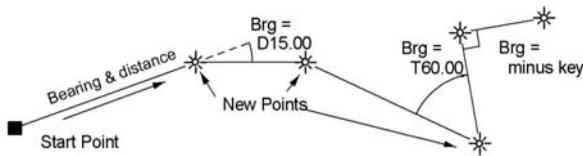


Figure 9-12. New Points and Lines Using the Traverse Option

To use the Traverse command, do the following:

1. Click **Insert ▶ Line ▶ Traverse**.
2. Click to select the start point or enter the coordinates into the input bar (Figure 9-13).



Figure 9-13. Traverse – Input Bar

3. Press **Enter** to create the point.
4. Click to select the line end point or enter either the point number or the bearing and distance from the start point into the input bar.

5. Press **Enter** to create a line.


The end point of the newly created line will be selected by default as the start point of the next line. The Number, Bearing, and Distance input bar appears with the point number incremented (Figure 9-14).



Figure 9-14. Number, Bearing, and Distance Input Bar

6. Enter the number, bearing, and distance to locate the next point.
7. To end the option, press **Esc** or select **Cancel** from the right-click shortcut menu.

XYZ Line Use the XYZ command to create a line between points by identifying the coordinate position of each point. New points may be created and existing points can also be used. A traverse is created by repetition of this command because the end point of the last line becomes the start point for the next line by default.

When using the mouse to select existing points, the Snap Point command should be active by clicking **Settings ▶ Snap Mode** or by clicking the *Snap Nearest Point*  icon.

To use the XYZ command, do the following:

1. Click **Insert ▶ Line ▶ XYZ**.
2. Click to select the start point or enter the coordinates into the input bar (Figure 9-15).



Figure 9-15. XYZ – Input Bar

3. Press **Enter** to create the point.
4. For the next point, click to select the line end point or enter either the point number or the bearing and distance from the start point into the input bar.
5. Press **Enter** to create a line.

The end point of the last line is selected by default as the start point of the next line. The Number, Bearing, and Distance input bar appears with the point number incremented (Figure 9-16).

Number	48	Bearing	62.0000	Distance	6.3	Elev	52.0	Code	PT	<input checked="" type="checkbox"/> Contourable
--------	----	---------	---------	----------	-----	------	------	------	----	---

Figure 9-16. Number, Bearing and Distance Input Bar

- Enter the number, bearing, and distance to locate another point.
- To end the option, press **Esc** or select **Cancel** from the right-click shortcut menu.

Join Points Use the Join Points command to create a line by joining two points in sequence and repeating the option to create an unbroken series of lines traversing between points (Figure 9-17). Points are identified by either their point number or by selection with a mouse.

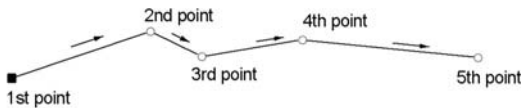


Figure 9-17. Join Several Points with Lines in a Sequence

To use the Joint Points command, do the following:

- Click **Insert ▶ Line ▶ Join Points**.
- Click the first point in the line or enter a **Start Point** number into the input bar (Figure 9-18).

Start Point	<input type="text"/>
-------------	----------------------

Figure 9-18. Join Points - Start Point

- Click the second point in the line or enter the **Next Number** into the input bar (Figure 9-19).

Next Point	<input type="text"/>
------------	----------------------

Figure 9-19. Join Points - Next Point

- Repeat Step 3 until the lines are joined.
- To end the command, press **Esc** or select **Cancel** from the right-click shortcut menu.

Join Two Points Use the Join Two Points command to create a line by joining two points. Points are identified by either their point number or by selection with a mouse.

To use the Joint Two Points command, do the following:

1. Click **Insert ▶ Line ▶ Join Two Points**.
2. Click the first point in the line or enter a **Start Point** number into the input bar (Figure 9-20).



Figure 9-20. Join Two Points - Start Point

3. Click the second point in the line or enter the **Next Number** into the input bar (Figure 9-21).



Figure 9-21. Join Two Points - Next Point

4. Repeat Step 3 until the lines are joined.
5. To end the command, press **Esc** or select **Cancel** from the right-click shortcut menu.

Perpendicular Line Use the Perpendicular command to create a new line perpendicular to a reference line from a nominated point or by offset and distance. The reference line may be an existing line, a line defined by any two points as a start point and an end point, or the line of a bearing from one point (Figure 9-22).

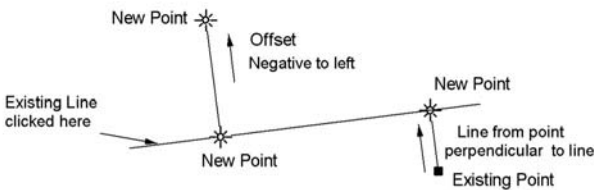




Figure 9-22. Two Options to Create Lines (and Points) Perpendicular to a Line

To use the Perpendicular command, do the following:

1. Click **Insert** ▶ **Line** ▶ **Perpendicular** or press the **Line Perpendicular**  icon on the **Add Entity** toolbar.



Make sure the **Snap Nearest Point**  icon is activated before choosing points with the mouse.

2. Select a line or line start point.
If a start point is selected, then a second point is required to define the line.
3. Select a second point or enter the number or bearing in the input bar on the lower left of the display area (Figure 9-23).

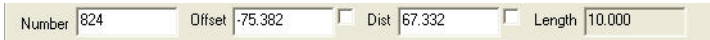


Figure 9-23. Perpendicular – Input Bar

4. Locate the position of the perpendicular by entering the offset from and the distance along the reference line, or click on an existing point.
5. Press **Enter** to create the perpendicular line and its two end points. Additional lines may be created perpendicular to the same entity.
6. To end the command, press **Esc** or select **Cancel** from the right-click shortcut menu.

Parallel Line Use the Parallel command to create a line that is parallel to and the same length as the selected line (Figure 9-24).

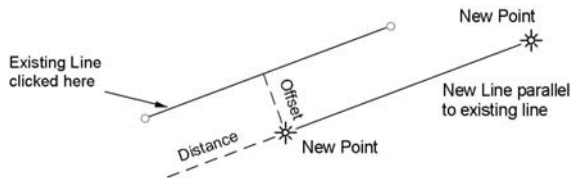




Figure 9-24. New Line Parallel to Existing Line

To use the Parallel command, do the following:

1. Click **Insert ▶ Line ▶ Parallel** or click the **Line Parallel**  icon on the Add Entity toolbar.



Make sure the **Snap Nearest Point**  icon is activated before choosing points with the mouse.

2. Select a line or line start point.
If a start point is selected, then a second point is required to define the line.
3. Select a second point or enter the point number in the input bar on the lower left of the display area (Figure 9-25).

When a second point is selected, the input bar displays again.



Figure 9-25. Parallel – input Bar

4. Locate the position of the parallel start point by clicking it or by entering the offset and distance into the input bar. The offset is the lateral displacement of the new line, positive right and negative left, and the distance is the linear displacement of the new line measured positive forward from the start point of the reference line. The start point is the point at the end of the line closest to the pointer when the line is selected using the mouse.
5. Press **Enter** to create the parallel line. Additional lines may be created parallel to the original reference line by repeating these steps.
6. To end the command, press **Esc** or select **Cancel** from the right-click shortcut menu.

Tangent Line Use the Tangent command to create a line tangential to an arc or circle from a selected point or from the end point of an arc (Figure 9-26 and Figure 9-27).

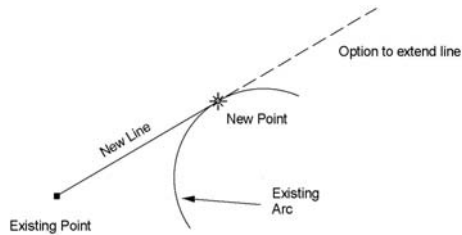


Figure 9-26. Lines as Tangent from an Existing Point to an Arc

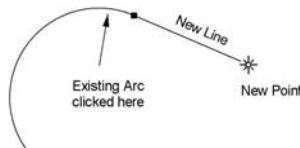


Figure 9-27. Line Created as Tangent from End of Arc

To use the Tangent command, do the following:


1. Click **Insert ▶ Line ▶ Tangent** or click the **Line Tangent**  icon on the Add Entity toolbar.
2. Select a point or an arc by doing one of the following:
 - Click a point from which the tangent line is defined.
 - Click an arc closest to the end from which the tangent is defined.
3. If a point is selected, then click to select the arc or circle and enter a length for the new line in the input bar on the lower left of the display area (Figure 9-28).



Figure 9-28. Point – Input Bar

The bearing from the start point to the tangent point (TP) is displayed, but the data box is grayed out and inaccessible. The distance data box displays the distance from the start point to the TP as default input.

4. Enter a distance or accept the default distance, and then press **Enter** to create the tangent.
5. If an arc is selected, the pointer is attached to the nearest end point of the arc.
6. Enter a length for the new line in the **Distance** field on the input bar or click to define a length (Figure 9-29).



Figure 9-29. Arc – Input Bar

7. Press **Enter** to create the tangent line and its two end points. Additional lines may be created perpendicular to the same entity.
8. To end the option, press **Esc** or select **Cancel** from the right-click shortcut menu.

External Tangent Line Use the External Tangent command to create a line between two arcs or circles as a tangent to both of them and terminating at the tangent points on the arcs or circles (Figure 9-30).

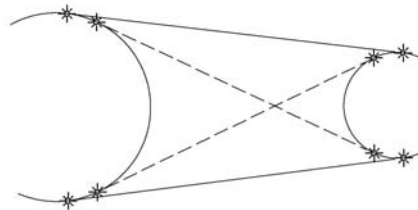



Figure 9-30. Four Solutions for External Tangents Between Two Arcs

To use the External Tangent command, do the following:

1. Click **Insert** ► **Line** ► **External Tangent** or click the **Line External Tangent**  icon on the **Add Entity** toolbar.
2. Select the first arc or circle at the approximate position of the tangent.
3. Select a second arc or circle at the approximate position of the tangent.

A tangential line is created between the two chosen entities with tangent points on the circumference of each arc or circle. The points do not segment the arc or circle.

There are four possible solutions. The solution created is determined by the pointer position when each of the arcs or circles is selected.

Splay Corner Command Use the Splay Corner command to create a splay corner between two intersecting lines (Figure 9-31) and (Figure 9-32).

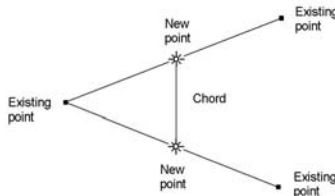


Figure 9-31. Splay Corner with no Truncation of Tangent Lines

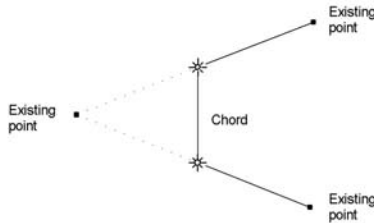


Figure 9-32. Splay Corner with Tangent Lines Truncated

To use the Splay Corner command, do the following:

1. Click **Insert** ► **Line** ► **Splay Corner** or click the *Splay Corner* icon on the **Add Entity** toolbar.
2. Select the first and second lines with the mouse.
3. Define the splay by entering either the **Chord Distance** or **Tangent Distance** in the input bar (Figure 9-33).



Figure 9-33. Splay Corner – Input Bar

4. Select the **Truncate Lines** check box to delete the two lines from the point of intersection to the splay; otherwise, they will be retained.

5. Press **Enter** to create the splay corner.
6. To end the command, press **Esc** or select **Cancel** from the right-click shortcut menu.

Note: You can also create the splay by dragging the chord into position and clicking with the left-mouse.

Fit Line Use the Fit Line command to create a straight line of the best fit through nominated points. A maximum offset can be specified and points outside this offset are excluded from the computation.

To use the Fit Line command, do the following:

1. Click **Insert ▶ Line ▶ Fit Line**.
2. Select the points in which you want to fit the line, and press **Esc**. The *Fit a Line* dialog box appears (Figure 9-34).

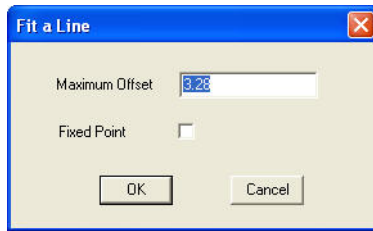


Figure 9-34. Fit a Line

3. Enter an offset to filter the points included in the computation.
4. Check mark the **Fixed Point** box to make the line go through a selected point. You are prompted to select this point with the pointer.
5. Press **OK**. A line is created that best fits the points and then a report is displayed.

Rectangle as Lines To use the Rectangle as Lines command, do the following

1. Click **Insert ▶ Rectangle as Lines**.
2. The **Rectangle as Lines** input bar appears (Figure 9-35 on page 9-19).

- Depending upon what action you are taking, you will see “Locate start point” or “Select width and height” display beneath the input bar.
- To end the command, press **Esc** or select **Cancel** from the right-click shortcut menu.



Figure 9-35. Rectangle as Lines

Define an Arc

An arc represents a curve between two points, which is a segment of the circumference of a circle. Its position depends on the points it joins and the arc center point. MAGNET Office also has a second arc entity called a 3-point arc. This type of arc uses three points on the arc circumference to define it. It has no center point.

The arc becomes a 3D arc if both points that define the arc have elevation values. Such an arc may be defined as a breakline for use in a surface model.

In This Section:

- “Angle” on page 9-19
- “Radius” on page 9-21
- “Three Points” on page 9-23
- “Tangent From Line” on page 9-25
- “TP-IP” on page 9-26
- “Fillet” on page 9-27
- “Chords” on page 9-28
- “Parallel” on page 9-29
- “Fit Arc” on page 9-30

Angle Use the Angle command to create an arc by locating the start point and center point of the arc, and then defining the arc by internal angle, chord length, or arc length (Figure 9-36 on page 9-20). The

start point is located by its coordinates, and the center point may be located either by coordinates or by bearing and distance from the start point according to the **Create Mode** selected from the **Settings** menu.

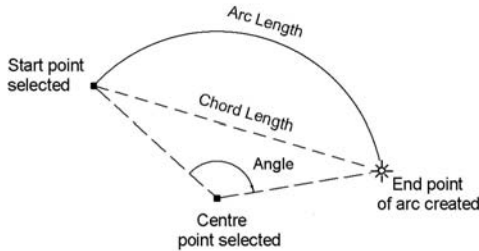



Figure 9-36. Arc Created from the Start and Center Point by Angle, Chord Length, or Arc Length

To use the Angle command, do the following:

1. Click **Insert** ► **Arc** ► **Angle**, or click the *Arc Angle*  icon on the **Add Entity** toolbar.
2. Click to select the start point of the arc with the mouse or enter either the point number or the coordinates into the input bar (Figure 9-37).

Number	1	East	518.212	North	100.931
--------	---	------	---------	-------	---------

Figure 9-37. Start Point – Input Bar

3. If the input mode is set to Coordinate Mode, then click the position of the center point or enter either the point number or the coordinates on the input bar.

If the input mode is set to Radiation Mode, then click the position of the center point as a bearing and distance from the start point or enter the coordinates on the input bar (Figure 9-38).

Number	832	Bearing	94.0863	Distance	34.939
--------	-----	---------	---------	----------	--------

Figure 9-38. Center Point – Input Bar

4. Locate the center point with the mouse or enter the bearing and distance from the start point into the input bar.
5. Press **Enter**.
6. Enter the internal angle, chord, or arc length in the input bar, and press **Enter** (Figure 9-39 on page 9-21).



Figure 9-39. Enter Internal Angle, Chord or Arc Length

7. Check mark the **Clockwise** box to measure the internal angle, chord length, and arc length in a clockwise direction. If not selected, the arc will be defined in an counter-clockwise direction.

The last point created becomes the start point for the construction of another arc.

8. To end the command, press **Esc** or select **Cancel** from the right-click shortcut menu.



To start a new arc from another start point, select **Reset** from the right-click menu.

Radius Use the Radius command to create an arc between two points by specifying a radius length (Figure 9-40 and Figure 9-41 on page 9-22). The two points may be defined by their coordinates or by the bearing and distance between them according to the Create Mode selected from the **Settings** menu.

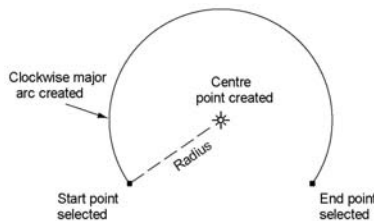


Figure 9-40. Clockwise Major Arc Created with Nominated Radius

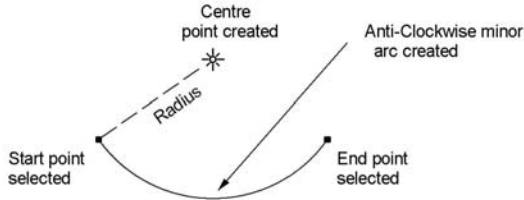


Figure 9-41. Anti-Clockwise Minor Arc Created with a Nominated Radius

To use the Radius command, do the following:


1. Click **Insert ▶ Arc ▶ Radius** or click the *Arc Radius*  icon on the **Add Entity** toolbar (Figure 9-42).
2. Locate the start point of the arc with the mouse or enter either the point number or the coordinates into the input bar.



Figure 9-42. Add Entity Toolbar

3. If the input mode is set to Coordinate Mode, then click the position of the end point or enter either the point number or the coordinates on the input bar.

If the input mode is set to Radiation Mode, then click the position of the end point as a bearing and distance from the start point or enter the coordinates on the input bar (Figure 9-43).



Figure 9-43. Point – Input Bar

4. Locate the end point with the mouse or enter the bearing and distance from the start point into the input bar.
5. Press **Enter**.
6. Use the pointer to position the arc center point or enter the radius in the input bar (Figure 9-44).



Figure 9-44. Radius – Input Bar

7. Check mark the **Clockwise** box to determine the direction of the arc from the start to end point.
8. Check mark the **Major** box to determine which of two possible arcs is created.
9. Press **Enter** to create the arc.
10. To end the command, press **Esc** or select **Cancel** from the right-click shortcut menu.

The last point created will become the start point for the construction of another arc.



To start a new arc from another start point, select **Reset** from the right-click menu.

Three Points Use the Three Points command to create an arc through three points. The three points may be identified by their coordinates or by the coordinates of the first point, and then the bearing and distance between the points according to the mode selected from the **Settings** menu (Figure 9-45).

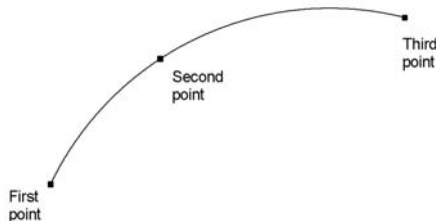


Figure 9-45. Arc Created from Three Points

To use the Three Points command, do the following:


1. Click **Insert** ► **Arc** ► **Three Points** or click the *Arc 3 Points*  icon on the **Add Entity** toolbar.
2. Click to select the start point of the arc or enter either the point number or the coordinates into the input bar (Figure 9-46).



Figure 9-46. Arc – Start Point

- If the input mode is set to Coordinate Mode, then click the position of the second point or enter either the point number or the coordinates on the input bar.
If the input mode is set to Radiation Mode, then click the position of the second point as a bearing and distance from the start point or enter the coordinates on the input bar (Figure 9-47).



Figure 9-47. Arc – Second Point

- Click to select the second point or enter the bearing and distance from the start point onto the input bar.
- Press **Enter**.
- If the input mode is set to Coordinate Mode, then click the position of the end point or enter either the point number or the coordinates on the input bar.
If the input mode is set to Radiation Mode, then click the position of the end point as a bearing and distance from the start point or enter the coordinates on the input bar (Figure 9-48).



Figure 9-48. Arc – End Point


- Click to select the end point or enter the point number or bearing and distance from the second point onto the input bar.
- Press **Enter** to create the arc.
- To end the command, press **Esc** or select **Cancel** from the right-click shortcut menu.

The last point created will become the start point for the construction of another arc.



To start a new arc from another start point, select **Reset** from the right-click menu.



To create a point at the arc center point, select the Snap Arc Center  icon and click the cursor on the required arc. If the arc is a 3-point-arc with no center point defined, then the arc center point is created.

Tangent From Line Use the Tangent From Line command to create an arc as a tangent extending from the end point of an existing line or arc (Figure 9-49) and Figure 9-50. This command has four possible solutions.

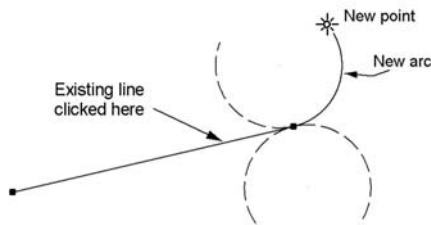


Figure 9-49. For Options to Create an Arc at a Tangent to the End of the Line

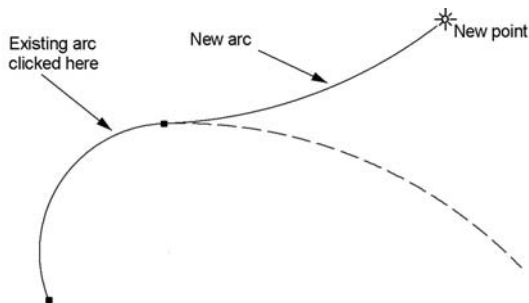


Figure 9-50. Options to Create an Arc Tangential to an Existing Arc

To use the Tangent command, do the following:

1. Click **Insert ▸ Arc ▸ Tangent From Line**.
2. Select a line, arc, or string.

The point at the end of the line closest to the pointer when the entity is selected becomes the start point.

3. Move the pointer to the approximate position of the arc center point, and enter the radius of the new arc in the input bar. You can enter the radius as a positive or negative number to define the direction of the offset of the new arc (Figure 9-51).

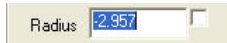


Figure 9-51. Arc – Radius

4. Enter the internal angle, chord, or arc length in the input bar (Figure 9-52).



Figure 9-52. Arc Angle, Chord, and Length

5. Select the **Clockwise** check box to determine the direction of the arc from the start to end point.
6. Press **Enter**.

TP-IP Use the TP-IP command to create an arc tangential to two lines or strings. The arc is defined by a radius or a tangent length. The tangent lines may be extended or truncated to fit the arc (Figure 9-53) and (Figure 9-54).

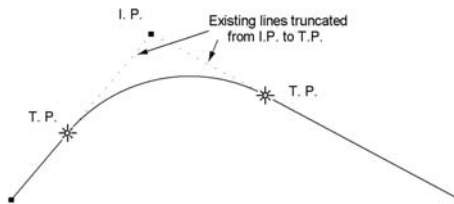


Figure 9-53. Arc Inserted Between Two Existing Lines. Lines are Truncated

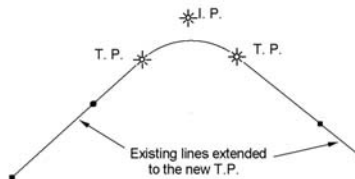


Figure 9-54. Two Existing Lines Extended to the New Tangent Points

To use the TP-IP command, do the following:


1. Click **Insert** ► **Arc** ► **TP-IP** or click the *Add TP-IP*  icon on the **Survey Tools** toolbar.
2. Click to select the first and second entity (line, arc, circle, or string).
3. Enter the **Radius** or the **Tangent Distance** for the intersection point of the selected entities to the arc tangent point (Figure 9-55).



Figure 9-55. Define Arc

4. Check mark the **Truncate Lines** box to delete the two lines from the intersection point to the tangent point; otherwise, they will be retained.
5. Press **Enter** to create the arc and related points.

Related points created in this option are the two tangent points (TP), the intersection point (IP), and the circle center (CC). The original lines as tangents to this arc are extended or truncated to fit the arc.

Fillet Use the Fillet command to create an arc as a fillet between any two entities – lines, arcs, circles, or strings. The arc is defined by a radius. The entities may be extended or truncated to the tangent points (Figure 9-56) and (Figure 9-57 on page 9-28).

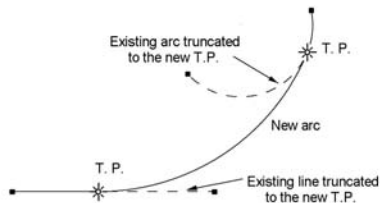


Figure 9-56. Arc Fillet Inserted Between Existing Line and Existing Arc: Arc and Line are Truncated

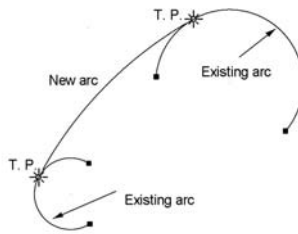


Figure 9-57. Arc Fillet Inserted Between Two Existing Arcs: No Truncation

To use the Fillet command, do the following:


1. Click **Insert** ► **Arc** ► **Fillet** or click the *Fillet Arc*  icon on the **Survey Tools** toolbar.
2. Click to select the first and second entity (line, arc, circle, or string).
3. Enter the **Radius** or click near the center of the arc fillet (Figure 9-58).



Figure 9-58. Arc Fillet

4. Select the **Truncate** check box to trim the original entities to the new tangent points; otherwise, they will be retained.
5. Press **Enter** to create the fillet arc.

Chords Use the Chords command to replace an arc with a series of chords either inside or outside of the arc. The chords created inside the arc will be of equal length. The chords created outside of the arc will be constructed as tangents of a specified number of equal length chords plus two half-length chords (Figure 9-59 on page 9-29).

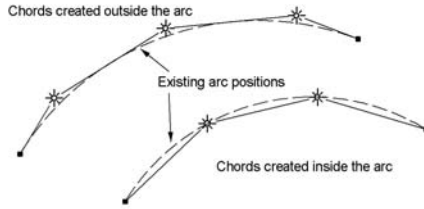


Figure 9-59. Arcs Converted to Chords

To use the Chords command, do the following:

1. Click **Insert ▸ Arc ▸ Chord**.
2. Select an arc or circle.
3. Select the number of chords required and their position inside or outside the arc (Figure 9-60).



Figure 9-60. Chord Number

4. Press **Enter** to convert the arc to chords with their associated points.

Parallel Use the Parallel command to create a parallel arc inside or outside the selected arc at a nominated offset from the selected arc (Figure 9-61).

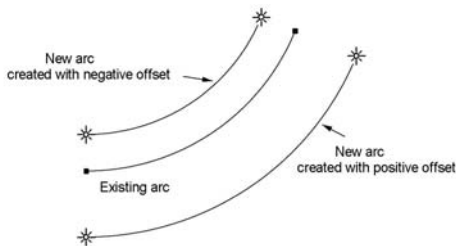


Figure 9-61. New Arcs Parallel to an Existing Arc – All Have a Common Center Point

To use the Parallel command, do the following:

1. Click **Insert ▸ Arc ▸ Parallel**.
2. Select an arc.

- Click on the arc position or enter an offset. A negative offset reduces the arc radius and creates it inside of the selected arc. A positive offset increases the arc radius and creates it outside of the selected area (Figure 9-62).

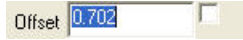


Figure 9-62. Offset Input Bar

- Press **Enter**.

Fit Arc Use the Fit Arc command to create an arc of the best fit through nominated points. A maximum offset can be specified and points outside this offset are excluded from the computation.

To use the Fit Arc command, do the following:

- Click **Insert ▶ Arc ▶ Fit Arc**.
- Select the points in which you want to fit the arc, and press **Esc**. The *Fit an Arc* dialog box appears (Figure 9-63).

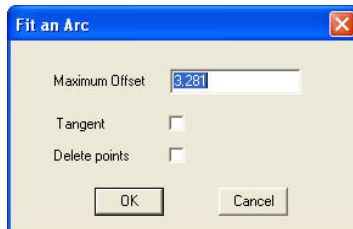


Figure 9-63. Fit an Arc

- Enter an offset to filter the points included in the computation.
- If the arc can be fitted tangent to two lines, check mark the **Tangent** box and click on the two lines when prompted.
- If you want to delete the points used to create the arc after it is created, select the **Delete** check box.
- Press **OK**.

The arc appears on the screen and then a report displays.

Define a Circle

A circle is similar to an arc but the start and end points are the same.

In This Section:

- “Radius”
- “Diameter” on page 9-32
- “A Circle Between Two Specified Points” on page 9-33

Radius Use the Radius command to create a circle by locating a center point and defining a radius (Figure 9-64).

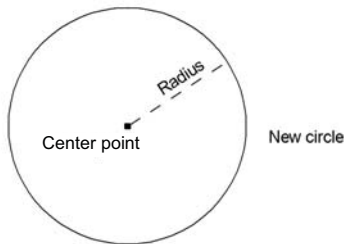


Figure 9-64. Circle Created from the Center Point and Radius

To use the Radius command, do the following:


1. Click **Insert** ▶ **Circle** ▶ **Radius** or click the *Arc Radius*  icon on the **Add Entity** toolbar.
2. Click to select the center point of the circle, or enter either the point number or the coordinates into the input bar, and press **Enter** (Figure 9-65).



Figure 9-65. Circle Radius Coordinates

3. Use the pointer to position the circle center point or enter the radius in the input bar, and press **Enter** (Figure 9-66).



Figure 9-66. Enter a Circle Point Radius

4. Check mark the **Clockwise** box to determine the direction of the circle from the start to end point.

5. Check mark the **Major** box to determine which of two possible circles is created.
6. Press **Enter** to create the circle.
7. To end the command, press **Esc** or select **Cancel** from the right-click shortcut menu.

Diameter Use the Diameter command to create a circle by locating two points, which will define its diameter (Figure 9-67). The two points are identified by either their coordinates or by the coordinates of the first point and the bearing and distance to the second point, depending on the Create Mode selected from the **Settings** menu.

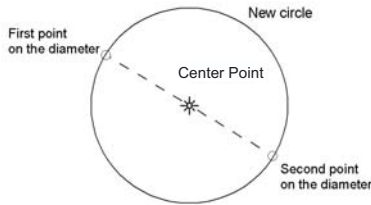


Figure 9-67. Circle Created from Two Points that Define the Diameter

To use the **Diameter** command, do the following:

1. Click **Insert ▶ Circle ▶ Diameter**.
2. Click to select the start point, or enter the point number, or the coordinates of the start point into the input bar, and press **Enter** (Figure 9-68).



Figure 9-68. Diameter – Input Bar

3. If the input mode is set to **Coordinate Mode**, then click the position of the diameter end point or enter either the point number or the coordinates on the input bar.

If the input mode is set to Radiation Mode, then click the position of the diameter end point as a bearing and distance from the start point or enter the coordinates on the input bar (Figure 9-69).



Figure 9-69. Diameter – Bearing and Distance

4. Click to select the next point, or enter the point number or bearing and distance from the start point into the input bar.
5. Press **Enter** to create the circle.

A Circle Between Two Specified Points The Points command creates a circle between two specified points. The points can be existing or you can create them by clicking on the screen.

Polylines

A polyline is a continuous line containing of a set of connected straight line segments. It is an open-shaped entity. The nodes of a polyline may be moved to change its shape.



If the polyline is deleted, the nodes (points) are retained.

To use the Polyline command, do the following:

1. Click **Insert ▶ Polyline**.
2. Locate the first point or line by doing one of the following:
 - Select a start point in the display area.
 - Enter values for the start point, and then press **Enter**.
3. Continue to locate points by clicking on them or entering values into the input bar until all of the points for the polyline are chosen.
4. Right-click, and select **Insert Polyline** from the shortcut menu.

Polygons

A polygon consists of several points in a sequence joined by lines and/or arcs to become an entity. The nodes of a polygon may be moved to change its shape. A polygon may be filled with a pattern, hatching, or shading. A polygon is the base entity for a lot, boundary, or pad.



If the polygon is deleted, the nodes (points) are retained.

The node points may be defined by their coordinates or by the bearing and distance between them according to the Create Mode selected from the **Settings** menu. An existing series of points joined by lines and/or arcs may also be converted into a polygon entity (Figure 9-70, Figure 9-71, and Figure 9-72).

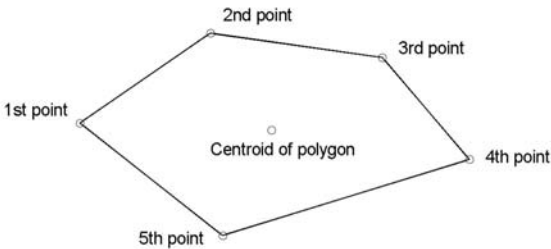


Figure 9-70. Polygon Defined from Five Points

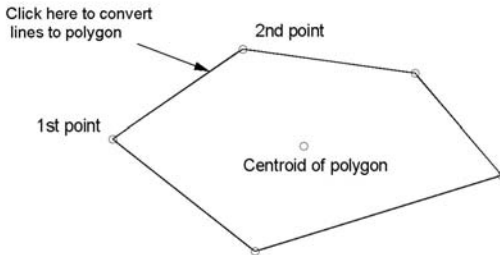
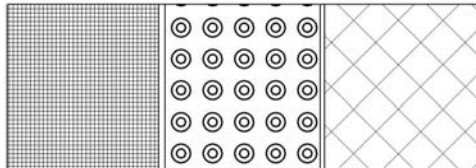



Figure 9-71. Polygon Created from Lines Defining a Closed Shape



Pattern Fill Symbol Fill Cross-

Figure 9-72. Fill Options for Polygons

To use the Polygon command, do the following:

1. Click **Insert ► Polygon** or click the *Create Polygon*  icon on the **Add Entity** toolbar.
2. Locate the first point or line by doing one of the following:
 - Selecting a start point and additional points and lines to close a polygon.

If the input mode is set to Coordinate Mode, then click the position of the next point or enter either the point number or the coordinates in the input bar (Figure 9-73).



Figure 9-73. Coordinate Mode – Input Bar

If the input mode is set to Radiation Mode, then click the position of the next point as a bearing and distance from the start point (Figure 9-74).



Figure 9-74. Radiation Mode – Input Bar

3. Continue to locate points until all of the points for the polygon are chosen.

When the start point is selected again, then the polygon is closed, and the **Edit Polygon** dialog box appears (Figure 9-75).

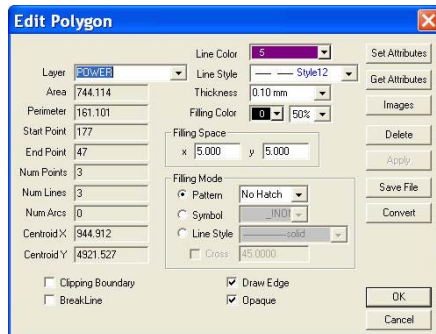


Figure 9-75. Edit Polygon Dialog Box

4. Confirm the properties, and click **OK** to save the polygon or click **Cancel** to quit the routine.

- Select an existing line to follow a path around the polygon shape.
 1. If a line is selected with the cursor, by default additional lines are highlighted along a followed path from the node nearest the clicked position. The highlight will run until:
 - The polygon is closed and the **Edit Polygon** dialog box appears.
 - The end of the available path is reached.
 - An intersection point is reached. If an intersection point is reached, then select the next line from the intersection point until the polygon definition is complete. Right-click, and select **Insert Polygon** from the shortcut menu.
 2. When the **Edit Polygon** dialog box appears, confirm the properties and click **OK** to save the polygon or **Cancel** to quit the routine.



A polygon can be converted to a lot, a boundary, or a pad. Click **Convert** on the **Edit Polygon** dialog box, select the new entity, and enter a name.



You can modify the polygon shape by moving its nodes. Select the polygon, press and hold the left **Shift** key, and drag a point to a new position.

Rectangle


A rectangle consists of at least four points and lines that join to become an entity. The points of a rectangle may be saved and/or moved to change its shape. You can fill a rectangle with a pattern, hatching, or shading. A rectangle is the base entity for a lot, boundary, or pad.



If the rectangle is deleted, the nodes (points) are retained.

The node points may be defined by their coordinates or by the bearing and distance between them according to the Create Mode selected from the **Settings** menu.

To use the Rectangle command, do the following:

1. Click **Insert ▶ Rectangle**, or click the *Insert Rectangle*  icon on the **Add Entity** toolbar.

The input bar appears.

2. Click on a start point to create one corner of the rectangle, as stated in the prompt line.

The **Save Points** parameter appears on the input bar.

3. To save the rectangle points, check mark the **Save Points** box before continuing. This parameter disappears after you select the next point (Figure 9-76).



Figure 9-76. Rectangle Input Bar

4. Move the pointer at a diagonal, and click on the end point to create the opposite end of the rectangle. Or, enter the height and width into the input bar.

Strings

A string consists of several points in a sequence joined by lines and/or arcs to become an entity. A string may be labelled with a user-defined code. The string will become a 3D string if all of the points that define it have elevation values. A string may also be defined as a breakline for use in a surface model.

In This Section:

- “String”
- “Parallel String” on page 9-40
- “Parallel Figure” on page 9-40

String

Use the String command to create a string, which comprises a series of points (nodes) joined by lines or arcs. The node points may be defined by their coordinates or by the bearing and distance between them according to the Create Mode selected from the **Settings** menu. An existing series of points joined by lines may also be converted into a string entity.

Note: To prevent nodes from being moved, set the points to Locked. If the string is deleted, the nodes are retained (Figure 9-77) and (Figure 9-78).

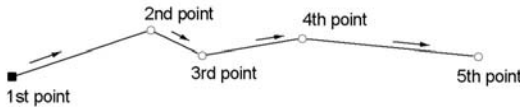


Figure 9-77. String Created from Points

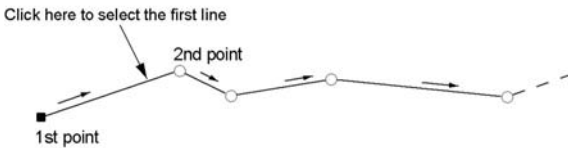



Figure 9-78. String Created from a Series of Connected Lines

To use the String command, do the following:

1. Click **Insert** ► **String** or click the *Create String*  icon on the **Add Entity** toolbar.
2. Locate the first point or line by doing one of the following:
 - Selecting a start point and additional points and lines to define a string and doing one of the following:
 - If the input mode is set to Coordinate Mode, then click the position of the next point or enter either the point number or the coordinates on the input bar (Figure 9-79).

Number	1	East	518.212	North	100.931
--------	---	------	---------	-------	---------

Figure 9-79. Coordinate Mode – Input Bar

- If the input mode is set to Radiation Mode, then click the position of the next point as a bearing and distance from the start point (Figure 9-80).

Number	832	Bearing	94.0863	<input checked="" type="checkbox"/>	Distance	34.939	<input checked="" type="checkbox"/>
--------	-----	---------	---------	-------------------------------------	----------	--------	-------------------------------------

Figure 9-80. Radiation Mode – Input Bar

- Continue to locate points until all of the points for the string are chosen, and skip to step 3.
 - Selecting a line close to the end point from which the string path will continue.
 - If a line is selected with the pointer, by default additional lines are highlighted along a followed path from the node nearest the clicked position. The highlight will run until the end of the path of joined lines is reached or until an intersection point is reached.
 - To continue the string beyond the intersection, select the next line from the intersection point until the string selection is complete.
3. Right-click and select **Insert String** from the shortcut menu. The *Create String* dialog box appears (Figure 9-81).

Figure 9-81. Create String Dialog Box

4. Confirm the properties, and click **OK** to save the string or click **Cancel** to quit the routine.



You can move the whole string by holding down the right **Shift** key and dragging the string.

Parallel String

To create a parallel string, do the following:

1. Click **Insert ▶ String ▶ Create Parallel String**.
2. Select a string, as stated in the input bar on the lower left side of the window.

The *Parallel String* dialog box appears (Figure 9-82).

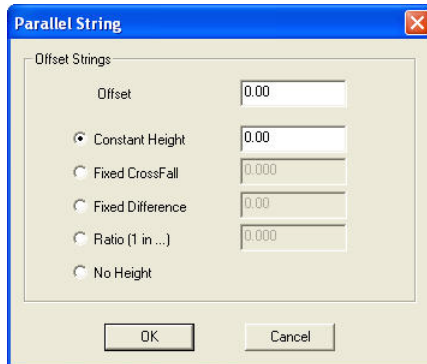


Figure 9-82. Parallel String Dialog Box

3. Enter an offset, and select the required preferences.
4. Click **OK** and the string appears in the display area according to the criteria you selected.



After clicking OK you will be prompted for the Side of the Offset. The Edit String dialog box displays for optional property definitions. After clicking OK again, you have the option to repeat the offset by pressing either the Enter key or the Esc button to reset the offset information. Press Esc twice to end the command.

Parallel Figure

Use the Create Parallel Figure command to compute figures that are parallel to any existing figure. The new figure, comprising points,

lines and arcs, are created parallel to an existing figure of selected points, lines and arcs, or a string.

To create a parallel point, line, arc, string, or alignment, do the following:

1. Click **Insert ► String ► Create Parallel Figure**.
2. Select a point, line, arc, string, or alignment, as stated in the input bar on the lower left side of the window.
 - If you selected a string, then the **Parallel Figure** dialog box appears.
 - If you selected a line, use the line at the end of the pointer to follow a path along the existing figure, and then right-click and choose **Create Parallel Figure**.
 - If you selected a point, then continue selecting all points to define the figure. When the figure selection is complete, then right-click and choose **Create Parallel Figure** from the shortcut menu.

The *Parallel Figure* dialog box appears (Figure 9-83).

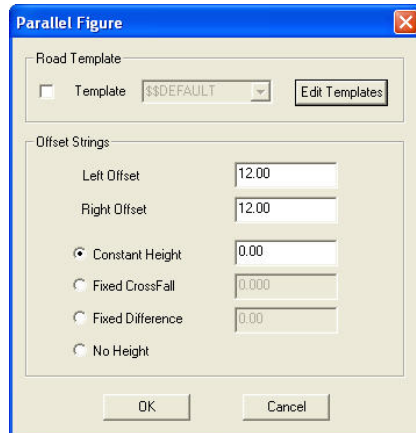


Figure 9-83. Parallel Figure Dialog Box

3. For points with elevations, you may want to use a template by check marking the **Template** box.

To edit or create a template:

1. Click **Edit Template**, and then click **New** to create a new template.
2. Designing the template from the center out, define the horizontal and vertical offsets for one side. Click either **CopyToRight** or **CopyToLeft**, depending on which side (left or right) you entered data, to copy the data to the opposite side.
3. Click **OK** to save.
4. If you are not using a template (The **Template** box is not checked), then enter an offset and select the required options. For option descriptions, see Table 9-1 below.
5. When you have finished defining a template or preferences, click **OK**, and the string appears in the display area according to the options you selected.

Shortcut Menu Options The shortcut menu options provide choices that help you complete your task. To use the shortcut menu options, right-click and select one of the following options:

- **Create Parallel Figure** – Opens the **Create Parallel Figure** dialog box, so you can define the parallel figure settings.
- **Back** – Goes back one step and removes the last line from the figure definition.
- **Undo** – Goes back one step and eliminates the last action performed.
- **Select Path By Two Ends** – Changes the line selection mode, so each successive line or point required for the figure must be selected with the mouse.
- **Select Path From Near End** – Change the line selection mode back to automatic highlight of a path along existing lines.

Table 9-1. Parallel Figure Field Descriptions

Field Name	Field Description
Left Offset	Enter the offset distance of the new figure to the left of the existing figure. If no figure is required to the left, leave the value as 0.00.

Table 9-1. Parallel Figure Field Descriptions

Field Name	Field Description
Right Offset	Enter the offset distance of the new figure to the right of the existing figure. If no figure is required to the left, leave the value as 0.00.
Constant Height	Select this, and enter the required height (elevation) for all the points created on the new figure(s).
Fixed CrossFall	Select this, and enter a crossfall. A height (elevation) is computed for all of the points created on the new figure(s). The crossfall is used to compute the height on a new point from the existing height of the corresponding point on the existing figure.
Fixed Difference	Select this, and enter a positive or negative value for the difference in height (elevation). A height (elevation) is computed for all the points created on the new figure(s). The height difference is used to compute the height on a new point from the existing height of the corresponding point on the existing figure.
No Height	Select this so that no height (elevation) is set on the points created on the new figure(s).

Define Text

Text is a string of alphanumeric data written at a particular insertion coordinate. You can enter or paste text into a project.

In This Section:

- “Insert Text” on page 9-44
- “Multiline Text” on page 9-46
- “Bearing and Distance” on page 9-47
- “Point Coordinate” on page 9-48
- “Point Height” on page 9-48
- “Point Offset” on page 9-49
- “Arc Length” on page 9-50
- “Arc Radius” on page 9-51

- “Stacked Arc Labels” on page 9-51
- “Angle” on page 9-52
- “Custom Tables” on page 9-53

Insert Text

The text string may be filled by data relating to a particular entity. It can also be linked to a point or line, so it is automatically removed if the point or line is deleted (Figure 9-84) and (Figure 9-85).

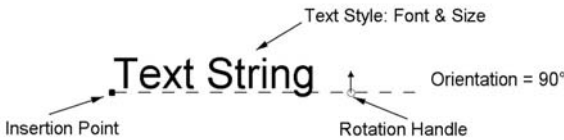


Figure 9-84. Default Insertion Style and Orientation

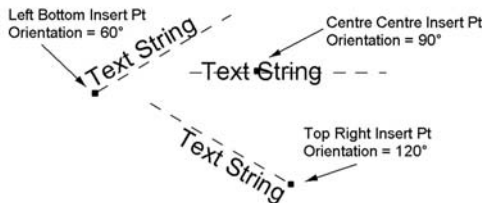


Figure 9-85. Different Insertion Styles and Orientation

To use the Insert Text command, do the following:

1. Click **Insert** ▶ **Text** ▶ **Text** or click the *Insert Text* **T** icon on the **Add Entity** toolbar.
2. Position the pointer where you want to input the text or type the coordinates into the input bar (Figure 9-86).



Figure 9-86. Text Input Bar

3. Type the text.

- On the Text Toolbar, select the correct text font, size, and style (Figure 9-87).

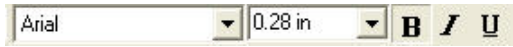


Figure 9-87. Text Toolbar

- On the Text input bar, click to select the **Insertion** point position. The insertion point relates to the held coordinate, relative to the text string. The drag handle for moving the text will be at the insertion position (Figure 9-88).

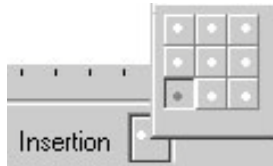


Figure 9-88. Insertion Point Selection

- To accept the text, press **Enter** or right-click and select **Accept Text** from the shortcut menu.

Bearing or Angle The default bearing or angle displayed on the input bar will position the text horizontally. The value applied for the orientation depends on the **Rotation By** selection in **Settings ▶ Project Settings ▶ Unit Settings**.

When **Rotation By** is set to **Survey Bearing**, the horizontal angle is 90.0000 degrees (similar to the bearing of a horizontal line and measured in a clockwise direction). This is the recommended setting as the values for the orienting of the text parallel to a line will match the bearing value of the line.

When **Rotation By** is set to **Mathematical Angle**, the horizontal angle is 0.0000 degrees (along the X axis and measured in a counter-clockwise direction), similar to CAD format text in other software.

Additional Information:

- You can move the text string by dragging the text to a new position with the pointer
- You can rotate the text string using the arrow handle at the right of the string.

- You can copy text by selecting it and dragging it while pressing **Ctrl**.
- You can insert text from the Microsoft Windows clipboard. Position the cursor at the insertion point, and click **Edit ▶ Paste**.

Multiline Text

Use the Text Box command to enter multiline text within the limits of a specified box boundary.

To use the Text Box command, do the following:

1. Click **Insert ▶ Text ▶ Text Box**.
The text box becomes attached to the pointer.
2. Click on the start point where you want the text to appear, and click on a second point at an opposite angle to create the box.
3. Type some text, and then double-click on it.
The *Edit Multiline Text* dialog box appears (Figure 9-89).

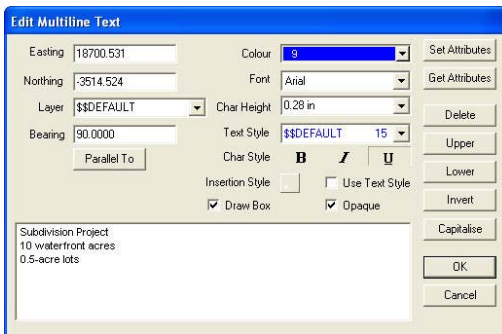


Figure 9-89. Edit Multiline Text

4. Finish typing your text in the text window and define any font settings as required.

Table 9-2 below describes the edit multiline text features.

Table 9-2. Edit Multiline Text Feature Descriptions

Feature Name	Feature Description
Set Attributes	Sets the current project settings to the settings of this text.
Get Attributes	Sets this text to the current project settings.

Table 9-2. Edit Multiline Text Feature Descriptions

Feature Name	Feature Description
Layer	The specified text layer
Bearing	The specified rotation of the text in bearing format.
Parallel To	Create text parallel to the selected text, line, or points.
Use Text Style	Forces the attributes to the text style.
Delete	Deletes the entire text box entry.
Upper	Changes the selected text to upper case.
Lower	Changes the selected text to lower case.
Invert	Inverts the selection. The lower case selections become upper case and vice versa.
Capitalize	Changes the selection to sentence case, where the first letter of the first word in a sentence is capitalized.

Bearing and Distance

Use the Bearing and Distance command to write the bearing and distance of the selected line as two text strings (Figure 9-90).

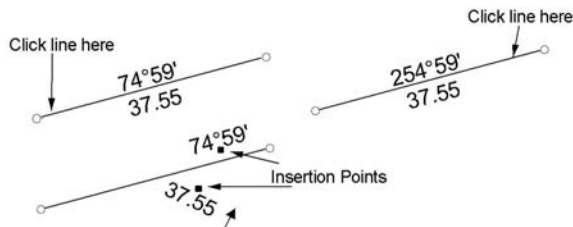


Figure 9-90. Bearing and Distance Written as Text

To use the Bearing and Distance command, do the following:

1. Click **Insert** ▶ **Text** ▶ **Bearing and Distance**.
2. Select a line or two points.
3. If you selected a point, then click on another point.
4. If you selected a line, then the bearing and distance of the line is read from the database and is written as text beside the line.
The text style, bearing and distance display, format and rounding are taken from the current style in the Line Annotation Settings.
5. Use the pointer to move, edit, or rotate the text.

Note: If the line is modified, then the bearing and distance text is not updated. To enable this feature, use Annotation instead of Text, and turn on the annotation flag for the line(s).

Point Coordinate

Use the Point Coordinate command to write the X and Y coordinates of a nominated point as two text strings (Figure 9-91).

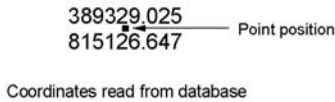


Figure 9-91. Coordinates Written as Text

To use the Point Coordinate command, do the following:

1. Click **Insert** ▶ **Text** ▶ **Point Coordinate**.
2. Select a point.

The point coordinates are read from the database and are written as text centered on the point.

3. Use the pointer to move, edit, or rotate the text as necessary. The text style, format and rounding are taken from the current style in the Distance Annotation Table.



If the point position is modified, then the coordinate is not updated.

Point Height

Use the Point Height command to write the height or elevation of a nominated point as a text strings (Figure 9-92).

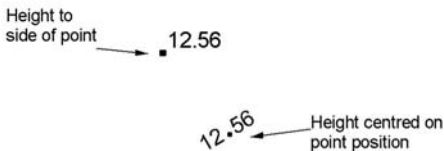


Figure 9-92. Coordinates Written as Text

To use the Point Height command, do the following:

1. Click **Insert ▶ Text ▶ Point Height**.
2. Select a point.
The point height is read from the database and is written as text beside the point or centered on the point.
3. Use the pointer to move, edit, or rotate the text as necessary.
The text style, height display, angle, and justification are taken from the current style in the Point Annotation Settings.



If the point height is modified, then the height text is not updated. To enable this feature, use Annotation instead of Text and turn on the Height Annotation flag for the point(s).

Point Offset Use the Point Offset command to write a dimension displaying the offset of a point perpendicular from a nominated line or offset from another point, complete with a dimension arrow. This command does not apply to polylines (Figure 9-93).

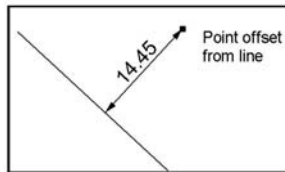


Figure 9-93. Offset Distance Written as Text

To use the Point Offset command, do the following:

1. Click **Insert ▶ Text ▶ Point Offset**.
2. Click to select a point.
3. Click to select a line or another point.

If you selected a line, the dimension is perpendicular from the point to the line.

If you selected a point, the dimension is between the two points; the text and arrow may be moved with the pointer.

The text style, offset display, rounding and the arrow size are taken from the current setting for dimensions in the Arrow and Dimension settings.



If the point(s) and/or line are moved, the dimension and offset text are not updated.

Arc Length

Use the Arc Length command to write the length of a nominated arc as a text string (Figure 9-94).

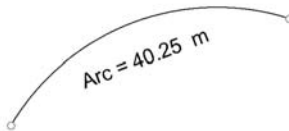


Figure 9-94. Arc Length Written as Text

To use the Arc Length command, do the following:

1. Click **Insert ▶ Text ▶ Arc Length**.
2. Select an arc.

The arc length is written as text centered half way along the arc; the text and arrow may be moved with the cursor.

The text style, arc length display, rounding and before/after text are taken from the current setting for arcs in the Arc Annotation Settings.



If the arc is modified, the arc length text is not updated.

Arc Radius

Use the Arc Radius command to write the radius of a nominated arc as a text string (Figure 9-95).

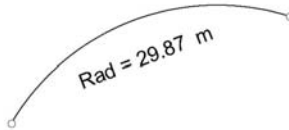


Figure 9-95. Arc Radius Written as Text

To use the Arc Radius command, do the following:

1. Click **Insert ▶ Text ▶ Arc Radius**.
2. Select an arc.

The radius is written as text centered half way along the arc; the text and arrow may be moved with the pointer.

The text style, arc radius display, rounding and before/after text are taken from the current setting for arc radius in the Arc Annotation Settings.



If the arc is modified, the arc text is not updated.

Stacked Arc Labels

Use the Stacked Arc Labels command to label arc data in stacked format. Content and order are set in the **Annotation Settings** dialog box (**Settings ▶ Annotation Settings ▶ Arc Annotation Settings**). If the arc is modified, the arc text is not updated.

To use the Stacked Arc Labels command, do the following:

1. Click **Insert ▶ Text ▶ Stacked Arc Labels**.
2. Select an arc.

The stacked labels appears inside the arc (Figure 9-96 on page 9-52).

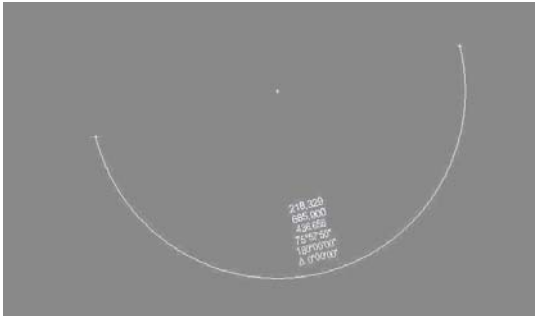


Figure 9-96. Stacked Labeled Arc

3. Select additional arcs as needed.

Angle

Use the Angle command to write the angle between two nominated lines or boundary edges of lots as text string (Figure 9-97).

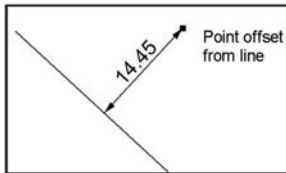


Figure 9-97. Angle Between Two Lines Written as Text

To use the Angle command, do the following:

1. Click **Insert** ▶ **Text** ▶ **Angle**.
2. Select the first and second line to define the angle or click on three points to define the angle.
3. The minor angle is measured between the two selected lines or the three points, and the text is written at the half angle between the lines. The text and arrow may be moved with the pointer. The text style, angle display, and rounding are taken from the current setting for dimensions in the Angle Annotation Settings.



If the lines are modified, the angle text is not updated.

Custom Tables

Click **Insert ▶ Table** to create custom tables (Figure 9-98). All tables are editable and are created directly from the data contained in the current project; however, they are not directly linked to the data in the project once they are created.

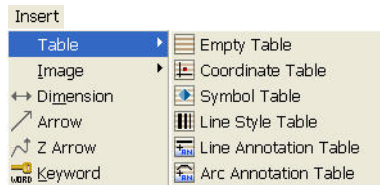


Figure 9-98. Create Custom Tables

You can insert the following types of tables:

- “Empty Table”
- “Coordinate Table” on page 9-54
- “Insert a Symbol Table” on page 9-55
- “Line Style Table” on page 9-57
- “Line Annotation Table” on page 9-59
- “Arc Annotation Table” on page 9-60

Empty Table To insert an empty table, do the following:

1. Click **Insert ▶ Table ▶ Empty Table** (Figure 9-99 on page 9-54).
2. On the *Table Properties* dialog box, select or enter the following information:
 - Name – select a table name.
 - Rows/Columns – enter the number of rows and columns desired for the table.
 - Color – select a color for the table from the drop-down entry box.
 - Font/Char. Height – select the font type and font size (height) from the drop-down entry box.
 - Text Style – enter a text style from the drop-down entry box

- Char Style – to select a character style, press either B (boldfaced), I (italicized), or (underlined) for table text character style.
- Insertion Style – choose a justification style: top left, top center, top right, middle left, middle center, middle right, bottom left, bottom center, or bottom right.
- Use Text Style – check mark this box to use the current style identified in the text properties toolbar.
- Opaque – check mark this box to make the text opaque, meaning that it will hide any part of an entity that it covers.
- Draw Box – check mark this box to draw a box around the table.

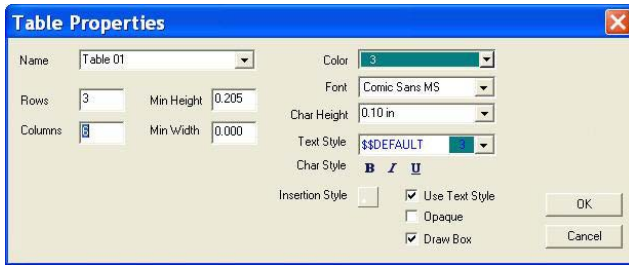


Figure 9-99. Table Properties – Empty Table



Until all cells are filled in with data, turning off the draw box option will NOT REMOVE the outer border.

Coordinate Table To insert a coordinate table, do the following:

1. Click **Insert ▶ Table ▶ Coordinate Table**.

The *Coordinate Table* dialog box appears (Figure 9-100 on page 9-55).

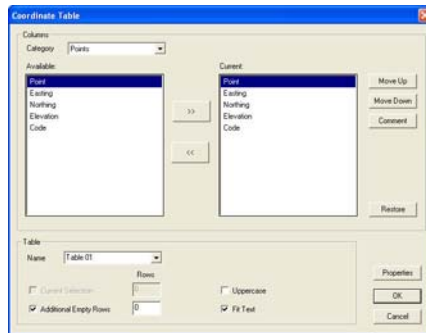


Figure 9-100. Coordinate Table

2. Select an entity from the **Category** menu to list in the table.
3. Customize the list order in the **Current** dialog box in one of two ways:
 - Select an item and click the left or right double-arrow icon to move that item to and from the **Available** and **Current** windows.
 - Select an item in the **Current** window and click the **Move Up** or **Move Down** button.
4. Make additional modifications as necessary. For field descriptions, see Table 9-3 below.
5. Click **OK** to accept the settings.

Table 9-3. Coordinate Table Field Descriptions

Field Name	Field Description
Comment	Adds a comment to the Current list.
Restore	Restores default column settings.
Properties	Opens the Table Properties dialog box.
Name	Displays the table name.
Additional Empty Rows	Adds additional rows to the table.
Uppercase	Sets all text to uppercase.
Fit Text	Automatically fits the text.

Insert a Symbol Table To insert a symbol table:

1. Click **Insert** ▶ **Table** ▶ **Symbol Table**.

The **Symbol Table** dialog box appears (Figure 9-101).



Figure 9-101. Symbol Table

2. Click in the first row of the **Symbol** column and select a symbol from the drop-down menu.
3. Type a description in the **Description** column.
4. Line styles and symbol tables can be built together by clicking **Append LineStyle**, which opens the **LineStyle Table** dialog box (Figure 9-102).

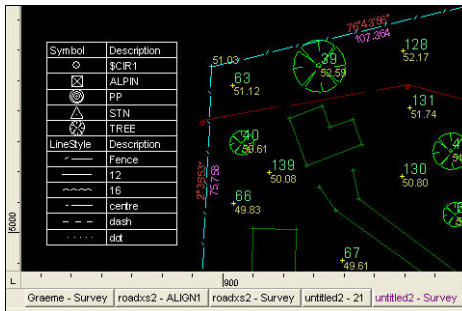


Figure 9-102. Example of Symbols and Line Styles, and Their Descriptions in the Same Table

5. Make additional modifications as necessary. For field descriptions, see Table 9-4.
6. Click **OK** to accept the settings.

Table 9-4. Symbol Table Field Descriptions

Field Name	Field Description
From Job	Inserts all symbols used from the current project.

Table 9-4. Symbol Table Field Descriptions

Field Name	Field Description
From Library	Inserts all symbols from the library.
Current Selection	Not available.
Restore	Restores default column settings.
Delete	Deletes selected row.
Name	Displays the table name.
Additional Empty Rows	Adds additional rows to the table.
Uppercase	Sets all text to uppercase.
Fit Text	Automatically fits the text.
Center Symbol	Places the symbols in the center of the cell, overriding the insertion style nominated.
Append Line Styles	Enables line styles to be included into the same table. This button opens the Line Styles Table dialog box.
Properties	Opens the Table Properties dialog box.

Line Style Table To insert a line style table:

1. Click **Insert ▶ Table ▶ Line Style Table**.

The *Line Style Table* dialog box appears (Figure 9-103).

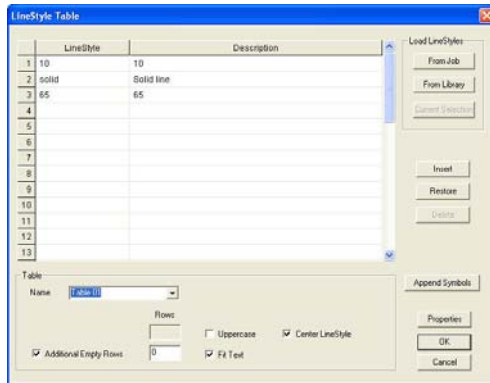


Figure 9-103. Line Style Table

2. Click in the first row of the *Line Style* column, and select a line style from the drop-down menu.
3. Type a description in the *Description* column.

- Line styles and symbol tables can be built together by clicking **Append Symbols**, which opens the *Symbol Table* dialog box (Figure 9-104).

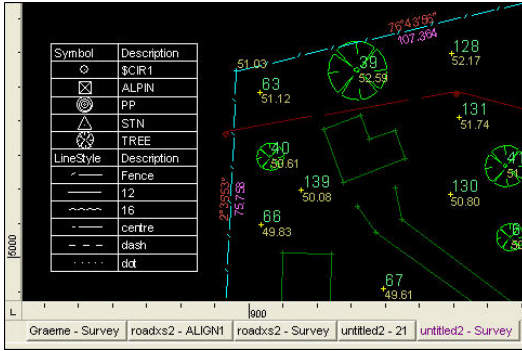


Figure 9-104. Example of Symbols and Line Styles, and Their Descriptions in the Same Table

- Make additional modifications as necessary.
For field descriptions, see Table 9-5.
- Click **OK** to accept the settings.

Table 9-5. Line Styles Table Field Descriptions

Field Name	Field Description
From Job	Inserts all line styles used from the current project.
From Library	Inserts all line styles from the library.
Current Selection	Not available.
Restore	Restores default column settings.
Delete	Deletes selected row.
Name	Displays the table name.
Additional Empty Rows	Adds additional rows to the table.
Uppercase	Sets all text to uppercase.
Fit Text	Automatically fits the text.
Center LineStyle	Places the line styles in the center of the cell, overriding the insertion style nominated.
Append Symbols	Enables symbols to be included into the same table. This button opens the Symbol Table dialog box.
Properties	Opens the Table Properties dialog box.

Line Annotation Table To insert a coordinate table:

1. Select the lines you want to include in the table.
2. Click **Insert ► Table ► Line Annotation Table**.

The *Line Annotation Table* appears on the screen, displaying the coordinates of the selected lines (Figure 9-105).



Number	Bearing	Distance
37	S37°27'02"E	28.354
38	N61°26'55"E	15.138
39	S37°27'02"E	28.896
40	N63°09'10"E	15.216
41	N61°26'55"E	15.382
42	S37°27'02"E	28.431
43	N63°09'10"E	15.481
47	S37°27'02"E	28.431
48	N61°26'55"E	7.863
49	S24°16'28"E	27.739
50	N63°09'10"E	14.336

Figure 9-105. Line Annotation Table

3. To move the table, place the pointer over a corner handle while the table is highlighted (in yellow), click and drag the handle to a new position.
4. To modify the table, double-click inside the table to open the *Annotation Table Settings* dialog box (Figure 9-106).

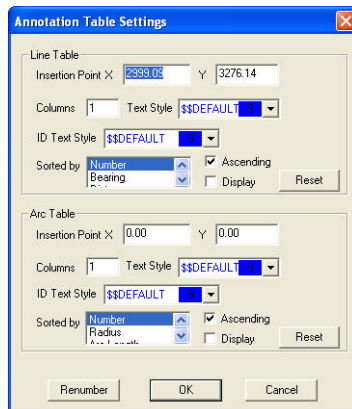


Figure 9-106. Annotation Table Settings

5. Make any necessary modifications in the **Line Table** box, and then click **OK** to save your changes.

Arc Annotation Table To insert an Arc Annotation table (Figure 9-107), do the following:

1. Select the arcs you want to include in the table.
2. Click **Insert ▶ Table ▶ Arc Annotation Table**.

The **Arc Annotation Table** appears on the screen, displaying the coordinates of the selected lines.

3. To move the table, place the pointer over a corner handle while the table is highlighted (in yellow), click and drag the handle to a new position.
4. To modify the table, double-click inside the table to open the **Annotation Table Settings** dialog box (Figure 9-107).

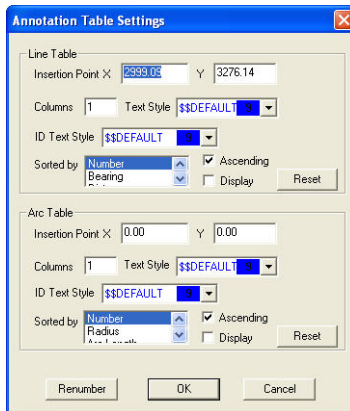


Figure 9-107. Annotation Table Settings

5. Make any necessary modifications in the **Arc Table** box, and then click **OK** to save your changes.

Image

An image that is inserted into the project is always kept in its original proportion and can be inserted as a background or part of the project. To insert an image successfully into a project, make sure it has one of these extensions: *.bmp*, *.jpeg*, *.jpg*, *.tiff*, *.tif*.

Images in MAGNET Office can be saved within the project or as a separate file that contains all of the project images. This enables you

to save projects with or without images, as some files can increase the final file size.

In This Section:

- “Insert Image” on page 9-61
- “Attach Image” on page 9-62
- “Insert Background Image” on page 9-62
- “Link/Unlink Image” on page 9-63
- “Viewing Attached/Linked Images to an Entity” on page 9-63

Insert Image To use the Insert Image command, do the following:

1. Click **Insert** ► **Image** ► **Insert Image**.
The **Open** dialog box appears.
2. Locate the folder in which the image file is saved, select the file, and then click **Open**.

The image is inserted into the project, enclosed in a yellow box with handles on three corners (Figure 9-108).

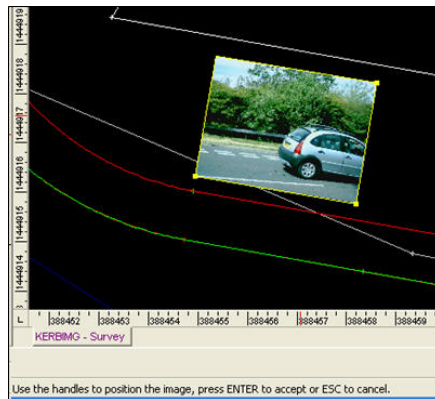


Figure 9-108. Inserted Image

3. To modify the size, pass the pointer over the upper right corner handle until it changes into a two-side arrow, and then move the corner handle inward to decrease the size or outward to increase the size of the image. Press **Enter** to accept the modification.

4. To move an image, pass the pointer over the bottom left corner handle until it changes into a four-side arrow and then move the image to a new position. Press **Enter** to accept the move.
5. To rotate an image, pass the pointer over the bottom right corner handle until it changes into a two-side arrow and then move the corner handle clockwise or counter-clockwise. Press **Enter** to accept the rotation.

Attach Image To attach an image to an entity:

1. Select the entity.
2. Click **Insert ▶ Image ▶ Attach Image to Entity**.
The *Open* dialog box appears.
3. Locate the folder in which the image file is saved, select the image file, and then click **Open**.
The image is attached to the entity.
4. To view the image, double-click on the entity to view its properties and then click **Images**. The attached image displays.
You can also select the entity, and click **Insert ▶ Image ▶ View Attached/Linked Images of an Entity**.

Insert Background Image Use the Insert Background Image command to insert a background image into the Survey View. Points on the image can be matched to points in the database.

To insert a background image, do the following:

1. Click **Insert ▶ Image ▶ Insert Background Image**.
The *Open* dialog box appears.
2. Locate the folder in which the image file is saved, select the image file, and then click **Open**.
The image is inserted into the project background, and the prompt line appears.
3. Select the first point to be matched with the image.
4. Select the point in the image to match the first point selected in the project.
5. Repeat steps 3 and 4 for the remaining three points.

6. Once the corresponding points on the images are set to the project points, right-click and select **Accept the Image**.

Note: A minimum of 2 and a maximum of 32 points is required for positioning the image into the project.

Link/Unlink Image Use the Link/Unlink Images to an Entity command to link an image to an entity, so the image is displayed in the drawing. If the entity the image is linked to is deleted, the image is also removed. The position of the image to the entity is maintained if the entity is moved, scaled, rotated, etc.

To link an image to an entity, do the following:

1. If it is not already inserted, then insert the image you want to link.
2. Select the entity.
3. Click **Insert ▶ Image ▶ Link/Unlink Images to Entity**.
4. Select the image, and then press **Esc**.
The image and the entity are linked. When one is selected so is the other.
5. To view the image, double-click on the entity to view its properties and then click **Images**. The attached image displays. You can also select the entity, and click **Insert ▶ Image ▶ View Attached/Linked Images of Entity**.

To unlink an image to an entity:

1. Select the entity.
The entity and the image become highlighted.
2. Click **Insert ▶ Image ▶ Link/Unlink Images to an Entity**.
3. Select the image.
The image is no longer highlighted and becomes unlinked to the entity.

Viewing Attached/Linked Images to an Entity Use the View Attached/Linked Images of an Entity command to view an image that has been attached or linked to an entity.

To use this command, select the entity, and then click **Insert ▶ View Attached/Linked Images of Entity**.

The image appears.

Dimension

Use the Dimension command to measure the distance between two points and write the dimension above or below a line that has arrowheads at each end (Figure 9-109).

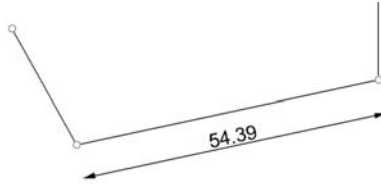



Figure 9-109. Dimension Arrow Annotated with Length of Line

To use the Dimension command, do the following:

1. Click **Insert ▶ Text ▶ Dimension** or click the **Create Dimension**  icon on the **Add Entity** toolbar.
2. Click on the starting point or select a line near its start point.
3. If you selected a point, then click on the end point.
The dimension is displayed between the two selected points or between the two points defining the selected line.
4. To move the dimension, right-click and select **Move Dimension** from the shortcut menu.
5. To edit the dimension, select the dimension and click **Modify ▶ Properties**.

The *Edit Dimension* dialog box appears (Figure 9-110).

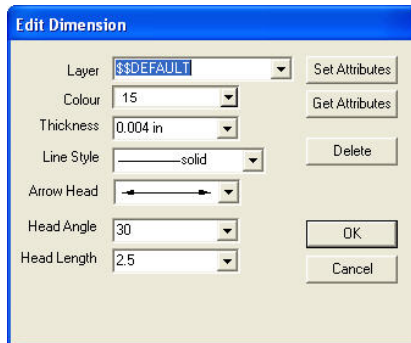


Figure 9-110. Edit Dimension Dialog Box

6. Modify the properties as required, and then click **OK** to accept the changes.

Insert Arrow

The arrow entity is a single line or series of lines that ends with an arrowhead. The arrow is used as an indicator for printed plans (Figure 9-111).

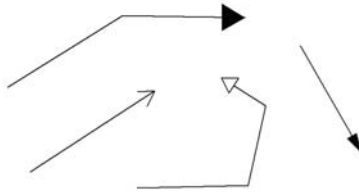


Figure 9-111. Various Options for Arrows

You can create either a Straight arrow or a Z arrow.

Straight Arrow

To use the Straight Arrow command, do the following:


1. Click **Insert ▶ Arrow** or click the **Create Arrow**  icon on the **Add Entity** toolbar.
2. Click on the starting point.
3. Click to define the end point of the line where the arrowhead will display.
4. To create the arrow, right-click and select **Insert Arrow** from the shortcut menu.
5. To modify the arrow style, select the arrow, and right-click to display the context (pop-up) menu and click **Properties**.
The *Edit Arrow* dialog box appears (Figure 9-112 on page 9-66).



Figure 9-112. Edit Arrow

6. Modify the properties as required, and then click **OK** to accept the changes.

Z Arrow

To use the Z Arrow command, do the following t:

1. Click **Insert** ▶ **Z Arrow**.
2. Click on the starting point.
3. Click on the end point of the line where the arrowhead will display.

The *Z Arrow* appears on screen (Figure 9-113).

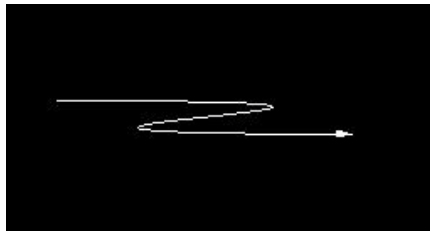


Figure 9-113. Z Arrow

Keywords

A keyword is one of eight words used in the design of a title block to represent data or entered text strings that relate to the individual project (Figure 9-114). Each keyword represents fields on the *Project Details* tab of the *Project Settings* dialog box.

A keyword is inserted into the title block design and replaced by the appropriate text string when the drawing is created.

TITLE = Keyword figure for manual
 Scale = 1:750
 Date = 24/07/2005

Figure 9-114. All Keywords Converted to Text Strings When Drawing Printed

To use the Keywords command, do the following:

1. Click **Insert ▶ Keyword**.

The *Keyword* dialog box appears, displaying a list of keywords (Figure 9-115).

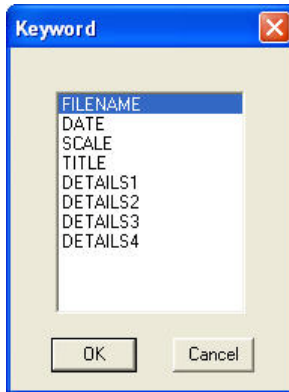


Figure 9-115. Keyword Window

2. Select the required keyword, and click **OK**.
 The keyword displays in yellow at the pointer position with the \$ sign prefix.

3. Enter the text coordinates, and select an **Insertion** icon on the input bar (Figure 9-116).



Figure 9-116. Keyword Input Bar

4. Left-click to position the keyword and enter the X and Y coordinates of the insertion point.
The keyword is written into the project. The text style is taken from the settings on the **Text** toolbar.

Additional Information

- The text string for the last five keywords on the list should be entered in the **Settings ▶ Project Settings ▶ Project Details** tab for each project.
- The other three keywords are taken from the project name, computer system date, and the scale of the plan when printed.
- The four keywords DETAILS1 to DETAILS4 can be given additional names using the **User defined labels** option in the **Settings ▶ Program Settings ▶ User Information** tab. This changes the headings for these four keywords on the Project Details to ensure the definition of these entries. The actual keywords used remain listed and displayed as *DETAILS1* to *DETAILS4*.
- To write a text string defined for a particular keyword directly into a project, use the **Insert ▶ Text ▶ Text** command.

Survey View: Modify Menu

This section describes the different Survey View menu commands on the Modify menu located on the main menu toolbar (Figure 10-1).

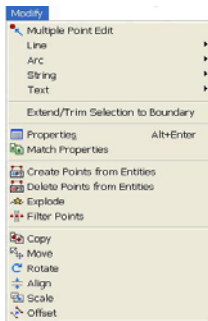


Figure 10-1. The Modify Menu

In This Section:

- “Edit Points” on page 10-2
- “Modify a Line” on page 10-6
- “Extend or Trim Line” on page 10-9
- “Extend All” on page 10-12
- “Trim All” on page 10-12
- “Multiple Trim” on page 10-10
- “Arc” on page 10-15
- “String” on page 10-19
- “Text” on page 10-27
- “Extend/Trim Selection to Boundary” on page 10-30
- “Properties” on page 10-31

- “Match Properties” on page 10-31
- “Create Points from Entities” on page 10-32
- “Delete Points from Entities” on page 10-32
- “Explode” on page 10-32
- “Filter Points” on page 10-33
- “Copy” on page 10-35
- “Move” on page 10-36
- “Rotate” on page 10-37
- “Align” on page 10-37
- “Scale” on page 10-40

Edit Points

The main entity in MAGNET Office is a point. A point represents a unique position and will have a coordinate (Easting, Northing, or X and Y value) to locate the point in terms of the datum or coordinate system of the dataset. It may also have an elevation (height or z value). Each point has a unique alphanumeric number to identify it in the MAGNET Office database.

In This Section:

- “Multiple Point Edit”
- “Set Point Height” on page 10-5

Multiple Point Edit

Click **Modify** ► **Multiple Point Edit** to edit more than one point at a time. The *Multiple Point Edit* dialog box appears (Figure 10-2 on page 10-3).

On the *Multiple Point Edit* dialog box, do the following to search for and find the points you want to edit and then to apply your modifications to those points.

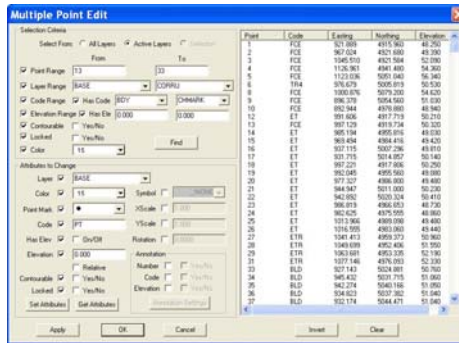


Figure 10-2. Multiple Point Edit

1. In the **Selection Criteria** panel, enter your search specifications, then click **Find**. Table 10-1 describes the Multiple Point Edit fields.

Table 10-1. Multiple Point Edit Field Descriptions

Field Name	Field Description
Selection Criteria	
Select From	<ul style="list-style-type: none"> • All Layers: Selects from all layers. • Active Layers: Selects from only the active (turned on) layers.
From/To	Chooses from points that were selected before this command was executed.
Point Range	Selects all points within the specified point range.
Layer Range	Selects all points within the specified alphabetical range.
Code Range	Selects all point within the specified alphabetical code range. Check mark the Has Code box to select only the point with the specified code.
Elevation Range	Selects all point within the specified elevation range. Check mark Has Ele to select only the points with the specified elevation.
Contourable	Check mark this box to select the points that are either set to contourable (Yes box) or not contourable (No box). Leave this check box empty to select points that are not contourable.

Table 10-1. Multiple Point Edit Field Descriptions (Continued)

Field Name	Field Description
Locked	Check mark this box so that point properties and attributes cannot be edited; locked points cannot be moved with the mouse.
Color	Selects the desired color.
Attributes to Change	
Layer	Check mark this box to change the layer name.
Color	Check mark this box to change the layer color.
Point Mark	Check mark this box to change the symbol for the point.
Code	Check mark this box to change the code name.
Has Elev	Check mark Has Ele to select only the points with the specified elevation.
Elevation	Check mark this box to enter a new elevation parameter.
Contourable	Check mark this box to select all points that are flagged for use when creating a surface.
Locked	Check mark this box so that point properties and attributes cannot be edited; locked points cannot be moved with the mouse.
Set Attributes	Press to set new attributes for the point.
Get Attributes	Press to get new attributes for the point.
Symbol	Check mark this box to select a new name for the symbol to be used for the point.
XScale/YScale	Check mark this box to change the X/Y scale.
Rotation	Check mark this box to enter a value for the rotation of the symbol.
Annotation	
Number	Check mark this box to edit point numbers in the <i>Point Annotation Settings</i> dialog box.
Code	Check mark this box to edit codes in the <i>Point Annotation Settings</i> dialog box.
Elevation	Check mark this box to edit point elevation in the <i>Point Annotation Settings</i> dialog box.
Annotation Settings	Press to display the <i>Point Annotation Settings</i> dialog box to set desired parameters.
Point Column	Point listing.

Table 10-1. Multiple Point Edit Field Descriptions (Continued)

Field Name	Field Description
Code Column	Code listing.
Easting Column	List of Easting parameters
Northing Column	List of Northing parameters
Elevation Column	List of elevation parameters
Apply	Applies modifications entered in the <i>Attributes to Change</i> pane.
OK	Closes the <i>Multiple Point Edit</i> dialog box and displays the point modifications on screen.
Cancel	Quits the routine.
Invert	Searches for the specified points and updates the display with the results.
Clear	Clears the selection in the display.

- In the *Attributes to Change* panel, enter your modifications and then click **Apply**.

Set Point Height

Use the Set Point Height command to set the point height of a given slope from a reference point of a known height.

To set point height, do the following:

- Click **Modify ▶ String ▶ Set Point Height**.
- Select the point for which the height will be set.
- Select the point of the known height as a reference. The height on the first point is computed from this height.
- In the input bar, enter the slope from the reference point to the first point chosen.

If the height on the point is less than the height on the reference point, then enter a negative slope (Figure 10-3).



Figure 10-3. Set Point Height Input Bar

- Press the **Enter** key.

The *Edit Point* dialog box appears, displaying the new height value in the *Elevation* field (Figure 10-4).

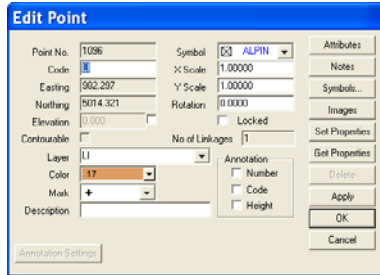


Figure 10-4. Edit Point Dialog Box



For a detailed description of the Edit Point dialog box, see “Defining a Point” on page 4-1.

6. Click **OK** to accept the height.

Additional Information If the first point chosen does not have the elevation selected, then double-click the point to display the *Edit Point* dialog box. Select the check box to the right of the grayed *Elevation* field. This activates this field and the height can be computed using this routine.

Modify a Line

The position of a line is totally dependent on the two points it joins. The line becomes a 3D line if both points that define it have elevation values. A line may also be defined as a breakline for use in a surface model.

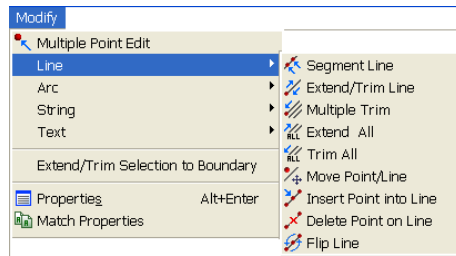


Figure 10-5. Modify – Line Menu Options

In This Section:

- “Segment Line” on page 10-7
- “Extend or Trim Line” on page 10-9
- “Multiple Trim” on page 10-10
- “Extend All” on page 10-12
- “Trim All” on page 10-12
- “Move Point/Line” on page 10-12
- “Insert Point Into Line” on page 10-13
- “Delete Point on Line” on page 10-14
- “Flip Line” on page 10-15

Segment Line

Use the Segment command to divide a line, arc, circle, or string into smaller segments, either by nominated distances or by equal parts from the start point of the entity. New points are created along the line, and if the start and end points have heights, then the height of each new point may be interpolated (Figure 10-6) and (Figure 10-7 on page 10-8).

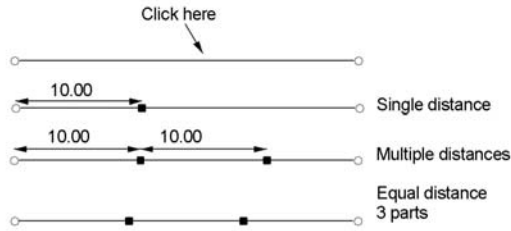


Figure 10-6. Segmented Lines

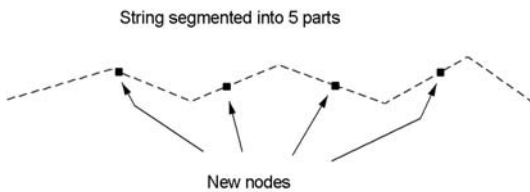



Figure 10-7. Segmented String

To use the Segment command, do the following:

1. Click **Modify** ► **Line** ► **Segment Line** or click the **Segment Line, Arc, or Circle**  icon on the **Survey Tools** toolbar (Figure 10-8).
2. Select a line, arc, circle, or string. If using a distance, click the entity near the end point from which the distance is to be measured.
3. Locate a point.

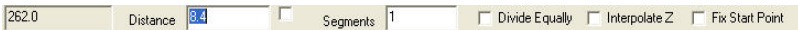


Figure 10-8. Segment Input Bar

The existing length of the entity is displayed at the left end of the input bar with various segment options. Select one of these options:

- **Distance** – Enter a value in the **Distance** field to segment the entity using a length from the beginning of the entity. A check mark automatically appears in the **Distance** check box to hold this value.

- Segments – Enter a value to create multiple segments.
 - Divide Equally – Select this check box to create segments of the same length.
 - Interpolate Z – Select this check box to compute a height on each new point.
4. Left-click to create the segment.
The last segment created will be highlighted for further segmenting.
 5. Click to accept additional segments.
 6. Press **Esc** to end the option or select **Cancel** from the right-click shortcut menu.

Extend or Trim Line

Use the Extend/Trim Line command to extend or trim an existing line, arc, string by a nominated distance, or an intersection with a selected line or arc. The selected entity will be lengthened or shortened (Figure 10-9).

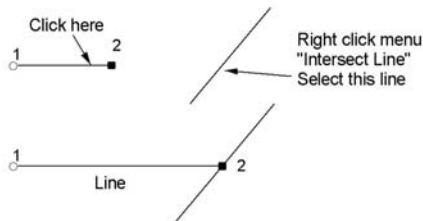



Figure 10-9. Extend Line to the Intersection with Selected Line

To use the Extend/Trim Line command, do the following:

1. Click **Modify** ▶ **Line** ▶ **Extend/Trim Line** or click the **Extend or Trim Line**  icon on the **Survey Tools** toolbar.
2. Select a line, arc, or string nearest the end that will be extended or trimmed.

The existing length of the entity is displayed in the input bar (Figure 10-10).



Figure 10-10. Extend/Trim Input Bar

3. To extend the trim line, do one of the following:
 - Drag the end of the entity with the pointer.
 - Enter a new length in the input bar, using the + sign to indicate a positive value and extend the entity or using the - sign to indicate a negative value and trim the selected entity.
 - Right-click and select **Intersect Line or Arc** from the shortcut menu, select the intersecting point, and then select again the entity to be extended.
 - Check mark the **Line/Arc Intersection** box to extend or trim the entity to its intersection point with another line or arc. Enter the **Offset** value and the **Adjust on Line** value or leave 0.0 in the fields for no offset or adjustment or click on the intersecting point, and then select again the entity to be extended (Figure 10-11).



Figure 10-11. Line/Arc Intersection Input Bar



If the point is locked, then the line cannot be extended or trimmed.

Multiple Trim

Use the Multiple Trim command to extend or trim several lines or arcs to their intersection with a selected line or arc. The selected entities will be lengthened or shortened. There is an additional option to insert the end point of the trimmed or extended lines or arcs into the reference line or arc (Figure 10-12 on page 10-11).



This command does not apply to polylines.

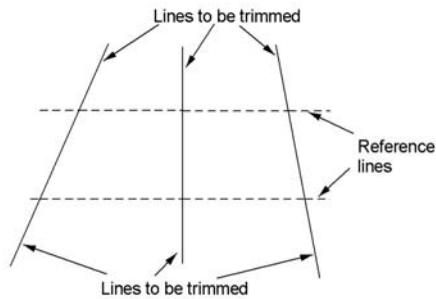


Figure 10-12. Reference Lines with Lines to be Trimmed

To use the Multiple Trim command, do the following:

1. Click **Modify** ► **Line** ► **Multiple Trim**.
2. Select a reference line or arc to which the entities will be trimmed or extended.
3. Define the offset in the input bar.
4. Click on a line or arc at the end that is to be extended or trimmed. The line or arc is connects to the reference line.
5. Click additional entities that you want to extend or trim.
6. When finished, right-click and select **Cancel** from the shortcut menu.

A confirmation window appears.

7. Click **Yes** to segment the reference line or arc with all of the new points.
8. Click **No** to complete the procedure without segmenting the reference line or arc (Figure 10-13).

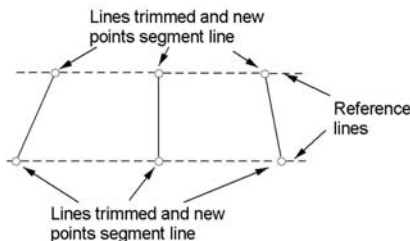


Figure 10-13. Trimmed Lines with New Points Segmenting Reference Lines

Extend All

Use the Extend All command to extend several existing lines, arcs, polylines or an intersection with a selected line or arc. The selected entities are lengthened.

To use the Extend All command, do the following:

1. Click **Modify ▶ Line ▶ Extend All**.
2. Select a line, arc, or polyline nearest the end that will be extended.

The lines, arcs, and/or polylines are extended to their intersection points with other lines or arcs.

3. Repeat Step 2 until you have finished, and then press **Esc**.

Trim All

Use the Trim All command to trim several existing lines, arcs, polylines or an intersection with a selected line or arc. The selected entities are shortened.

To use the Trim All command, do the following:

1. Click **Modify ▶ Line ▶ Extend All**.
2. Select a line, arc, or polyline nearest the end that will be shortened.

The lines, arcs, and/or polylines are trimmed to their intersection points with other lines or arcs.

3. Repeat Step 2 until you have finished, and then press **Esc**.

Move Point/Line

Use the Move Point/Line command to move a point with any attached lines or a line with both its defining points and any lines attached to these two points (Figure 10-14 on page 10-13).

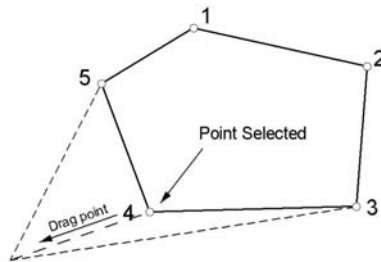


Figure 10-14. Point Dragged to a New Position with Lines Attached

To use the Move Point/Line command, do the following:

1. Click **Modify ▶ Line ▶ Move Point/Line**.
2. Select either a point or line that you want to move.
3. You can either drag the point or line to a new position as shown in Figure 10-14 or enter the bearing and distance in the input bar at the lower left of the display area.

Insert Point Into Line

Use the Insert a Point into a Line command to “snap” a point, with no links, into a line, arc, circle, or string, with a perpendicular move onto the entity. The line, arc, or circle is split into two entities. A point inserted into a string becomes a node in the string (Figure 10-15).

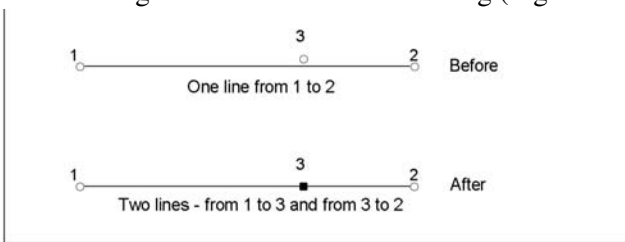



Figure 10-15. Point Three Moves into the Line and Segments It

To use the Insert a Point into a Line command, do the following:

1. Click **Modify ▶ Line ▶ Insert Point Into Line** or click the **Insert Point into Line**  icon on the **Survey Tools** toolbar.
2. Select a line, arc, circle, or string.

3. Select a point.

The point moves perpendicularly onto the line, arc, circle, or string. The line, arc, or circle is segmented into two lines or two arcs. The string will have a new node in the string definition.

Links If the point has a link, it will already be attached to another entity, such as a point defining a line and cannot be moved. Press **Ctrl** while selecting a new point to create.

- If the point is on the line, then it will be broken into the line.
- If the point is not on the line, a new point is created to segment the line, arc, circle, or string. This point is created in the same position that a point with no links would be moved to.

Locked Points If the point is locked, then the point cannot be moved. The lines are moved to the points instead of the point being moved.

Delete Point on Line

Use the Delete Point on Line command to delete a point that lies between two collinear lines or arcs, converting two lines to one line, or two arcs to one arc (Figure 10-16).

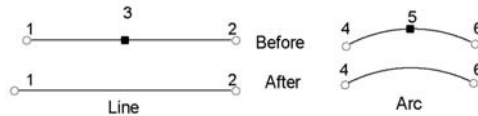



Figure 10-16. Point Removed and One Entity Formed

To use the Delete Point on Line command, do the following:

1. Click **Modify** ► **Line** ► **Delete Point on Line** or click the **Delete Connecting Point**  icon on the **Survey Tools** toolbar.
2. Click on the point to delete it.

If the lines joining the point are collinear, then the point will be removed and the two lines converted to one line.

If the arcs joining the point are collinear, then the arcs are converted to one three-point arc.

Flip Line

Use the Flip Line command to reverse a line horizontally. Click on the line you want to flip, then click **Modify ▶ Flip Line**.

Arc

An arc represents a curve between two points, which is a segment of the circumference of a circle (Figure 10-17)..

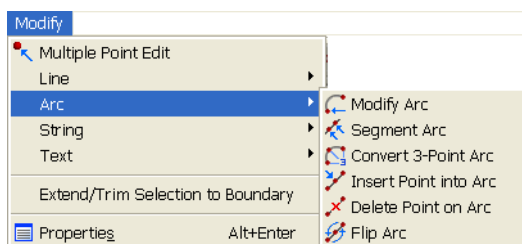


Figure 10-17. Modify – Arc Menu Option

In This Section:

- “Modify Arc”
- “Segment Arc” on page 10-16
- “Converting a 3-Point Arc” on page 10-16
- “Insert Point Into Arc” on page 10-18
- “Delete Point on Arc” on page 10-18
- “Flip Line” on page 10-18

Modify Arc

Use the Modify Arc command to modify existing arcs by changing the radius or the tangent distance.

To modify arcs, do the following:

1. Click **Modify ▶ Arc ▶ Modify Arc**.
2. Select an arc, as stated in the prompt line on the lower left side of the window.

The input bar appears. The **Radius** and **Tangent Distance** is displayed for reference (Figure 10-18).

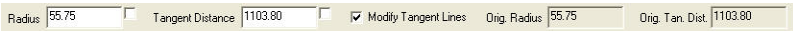


Figure 10-18. Modify Arc Input Bar



MAGNET Office automatically selects the **Modify Tangent Lines** check box so the tangent lines are updated to the new arc position.

3. Modify the arc by doing one of the following:
 - Enter a new **Radius** or **Tangent Distance** in the input bar. The arc automatically adjusts to the value(s) you entered.
 - Move the pointer over a yellow handle on the arc until it changes into a double-ended arrow. Depending on the direction the top arrow is facing, you can use the pointer to modify the arc radius and tangent distance.
4. When you are finished, press **Enter**.

Converting a 3-Point Arc MAGNET Office has two types of arcs. The first is created from two points and a center point. The second is created from three points (no center point used).

Use the Convert 3-Point Arc command to convert an arc defined through three points to an arc defined with two points (start and end) and a center point. When converting a 3-point arc, the third point is removed from the arc definition but not deleted.

To convert an arc, do the following:

1. Click the **Snap Arc Center**  icon.
2. Click the **Add a Point**  icon.
3. Click on the arc to create a center point.
The arc remains as a 3-point arc.
4. Right-click and select **Cancel** to end the routine.

Segment Arc

Use the Segment Arc command to divide an arc into smaller segments by nominated distances or by equal parts from the start

point of the entity. New points are created along the line and if the start and end points have heights, then the height of each new point may be interpolated.

To divide an arc into equal segments, do the following:


1. Select the arc you want to segment.
2. Click **Modify ▶ Arc ▶ Segment Arc** or click the *Segment Line*, *Arc*, or *Circle*  icon.
3. In the input bar, enter the number of segments in the **Segments** field. MAGNET Office automatically places a check mark in the **Divide Equally** box (Figure 10-19).



Figure 10-19. Arc Segment Input Bar

4. Press **Enter** for each segment you want to save.

To divide an arc into points you select, do the following:

1. Click **Modify ▶ Arc ▶ Segment Arc**.
2. Select an arc, as stated in the prompt line in the lower left side of the window.
3. Click on the arc to add a segment point, and then press **Enter**.
4. Continue to add segment points as necessary, pressing **Enter** after selecting each point.

Convert Three Point Arc


An arc's position depends on the points it joins and the center point of the arc. MAGNET Office has a second arc entity called a 3 point arc. This type of arc uses three points on the arc circumference to define it. It has no center point

1. Click **Modify ▶ Arc ▶ Convert 3-Point Arc**.

Insert Point Into Arc

Use the Insert Point Into Arc command to “snap” a point into an arc, segmenting the entity.


To snap a point into an arc, do the following:

1. Click **Modify ▶ Arc ▶ Insert Point Into Arc** or click the **Insert Point Into Line**  icon.
2. Select the arc, and then click the point you want to insert into the arc.
The point snaps into the arc.
3. Select another point, as stated in the prompt line, or right-click and select **Cancel** to end the routine.

Delete Point on Arc

Use the Delete Point on Arc command to delete a point on an arc, converting two arcs to one.

To delete a point, do the following:

1. Click **Modify ▶ Arc ▶ Delete Point on Arc** or click the **Delete Connecting Point**  icon.
2. Click the point to delete it.
If the arcs joining the point are collinear, then the arcs are converted to one 3-point arc.
If the point cannot be removed, then a message is displayed.

Flip Line

Use the Flip Line command to reverse a line horizontally. Click on the line you want to flip, then click **Modify ▶ Flip Line**.

String

A string consists of several points in a sequence joined by lines and/or arcs to become an entity. The string will become a 3D string if all of the points that define it have elevation values. A string may also be defined as a breakline for use in a DTM (Figure 10-20).

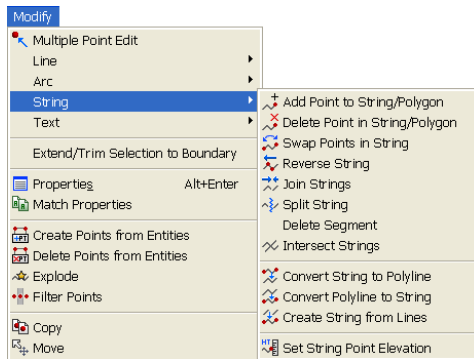


Figure 10-20. Modify String Menu Options

In This Section:

- “Add Point to String/Polygon” on page 10-20
- “Delete Point in String/Polygon” on page 10-20
- “Swap Points in String” on page 10-21
- “Reverse String” on page 10-21
- “Join Strings” on page 10-22
- “Split String” on page 10-23
- “Delete Segment” on page 10-23
- “Intersect Strings” on page 10-24
- “Convert Polyline to String” on page 10-25
- “Convert Polyline to String” on page 10-25
- “Create String From Lines” on page 10-25
- “Set String Point Elevation” on page 10-26

Add Point to String/Polygon

Use the Add Point to String/Polygon command to insert a point into a string or polygon, so the point becomes a node on the string/polygon. The string/polygon is moved to the point position (Figure 10-21).

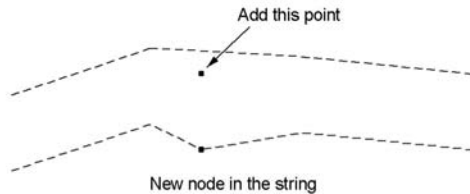


Figure 10-21. Add a Point to the String

To add points to a string/polygon, do the following:

1. Click **Modify** ▶ **String** ▶ **Add Point to String/Polygon**.
2. Select a string/polygon.
3. Click on a point to become a node on the string/polygon.
The point will become a new node in the string/polygon, so the string joins the point. The point is not moved into the string.
4. To move a point into the existing position of a string, click **Modify** ▶ **Line** ▶ **Insert Point into Line**.

Delete Point in String/Polygon

Use the Delete Point in String/Polygon command to remove a node from a string/polygon. The string/polygon is updated, and the point is deleted from the database (Figure 10-22).

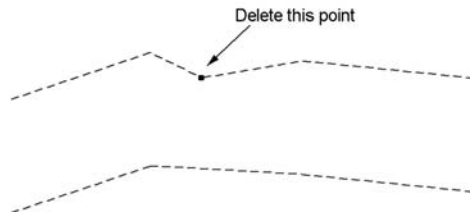


Figure 10-22. Deleting a Point

To remove a node from a string/polygon, do the following:

1. Click **Modify** ▶ **String** ▶ **Delete Point in String/polygon**.

2. Select a string/polygon.
3. Click on a point to delete it.

The point is removed from the string/polygon, and the string/polygon is reformed to the nodes on either side of the deleted point.

Swap Points in String

Use the Swap Points in String command to swap two adjacent nodes of a string. The definition of the string is updated to follow the new order of the nodes (Figure 10-23).

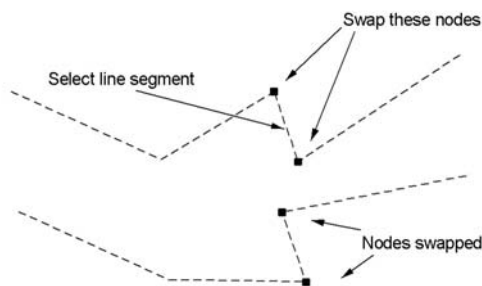


Figure 10-23. Order of Two Adjacent Nodes in a String Reversed

To swap two adjacent nodes of a string, do the following:

1. Click **Modify** ▶ **String** ▶ **Swap Points in String**.
2. Select a string.
3. Select a line segment on the string.

The two points at either end of the segment will be swapped, changing their order in the string.

Reverse String

Use the Reverse String command to define a string in the opposite direction of its current direction. This command shows one-directional line styles and can be used, for example, to show the flow of traffic or water (Figure 10-24 on page 10-22).

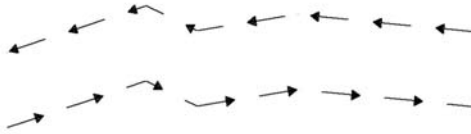


Figure 10-24. Reverse String

To use the Reverse String command, do the following:

1. Click **Modify ▶ String ▶ Reverse String**.
2. Select a string.

The string is reversed in direction, with the start point changed to the end point.

Join Strings

Use the Join Strings command to join two strings.

To join two strings, do the following:

1. Click **Modify ▶ String ▶ Join Strings**.
2. Select a string on the main screen.
3. Select a second string to join to the first one

The *Edit String* dialog box appears (Figure 10-25 on page 10-23).

4. Select the desired parameters for the string.
5. Press **OK**.

The two strings you selected will be joined on the main screen.

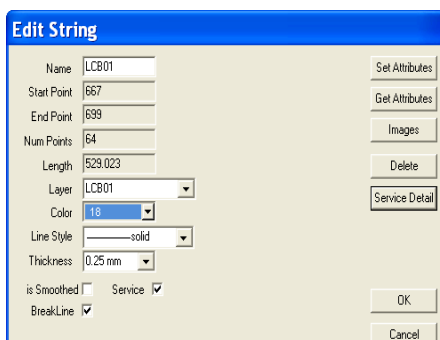


Figure 10-25. Modify – Join String Option

Split String

Use the Split String command to split a string into two at a selected point. This enables you to modify part of a string (Figure 10-26).

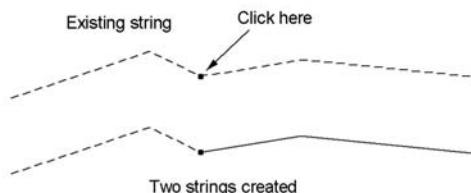


Figure 10-26. Existing String Split into Two at a Selected Point

To use the Split String command, do the following:

1. Click **Modify ▶ String ▶ Split String**.
2. Select a string.
3. Select a point on the string where you want it to split.
Two strings are created, joined at the selected point.

Delete Segment

Use the Delete Segment command to remove an unwanted segment.

1. Click **Modify ▶ String ▶ Delete Segment**.
2. At the prompt, select a string to delete.

3. Double-click the left mouse button to display the *Edit Point* dialog box (Figure 10-4 on page 10-6).
4. Delete the desired segment and press **OK** to save.

Intersect Strings

Use the Intersect Strings command to insert a point at the intersection of two strings. The strings must physically cross one another. Each string is then converted into two separate strings. For example, a crossing of two strings would result in the creation of four separate strings with a single common point.

To use the Intersect String command, do the following:

1. Click **Modify ▶ String ▶ Intersect String**.
2. Select a string, as stated in the prompt line, and then select another string.
3. Make any changes to the attributes on the input bar or confirm the attributes by pressing **Enter**.
A point appears where the two strings intersect.

Convert String to Polyline

Use the Convert String to Polyline command to convert a string back to points and individual lines:

1. Click **Modify ▶ String ▶ Convert String to Polyline**.
2. Click on a string.

The entity is removed and replaced by points joined by lines.



If it is difficult to select a polygon, then select it first and then click the **Explode String/Polygon** command.



Boundaries are a form of polygon. If a boundary is selected, then a confirmation message may appear.

Convert Polyline to String

Use the Convert Polyline to String command to convert a polyline back into a string:

1. Click **Modify** ▶ **String** ▶ **Convert Polyline to String**.
2. Click on a polyline.

The entity is removed and replaced by lines joined by points.



If it is difficult to select a polygon, then select it first and then click the **Explode String/Polygon** command.



Boundaries are a form of polygon. If a boundary is selected, then a confirmation message may appear.

Create String From Lines

Use the create string from lines command to create strings from line segments and to automatically join all string segments together to form a continuous multi-segmented string (Figure 10-27 on page 10-26).



Prior to executing the command to create strings from lines, there must be existing lines with common endpoints.



If all segments are not joined, check for overlapping strings segments and verify that all selected entities are lines.

To use the create strings from lines command, first select the lines to convert on the main screen, then click **Modify** ▶ **String** ▶ **Create String from Lines**.

Before 4 Individual Line Segments



After, One multi segmented string.



Figure 10-27. Create Strings From Lines

Set String Point Elevation

Use the Set String Point Elevation command (**Modify ▶ String ▶ Set String Point Elevation**) to set the point elevations that make up the string. Because each point can be on different layers, the option to change the elevation on active layers is useful. If a point that is contained in a string is turned off, check mark the **Active Layer Only** box to retain the string point elevation while changing the point on active layers. When the **Contourable** box is check marked, the point is used in surface creation (Figure 10-28).

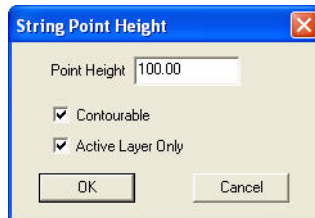


Figure 10-28. String Point Height

Text

Text is a string of alphanumeric data written at a particular insertion coordinate. The text has a defined color and style, such as font and character size.

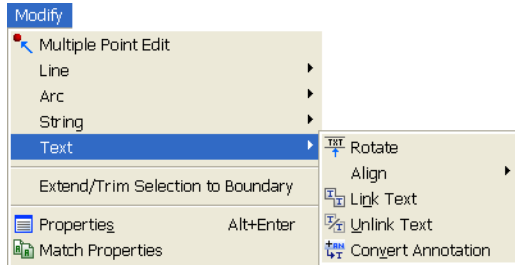


Figure 10-29. Modify – Text Menu Options

In This Section:

- “Rotate” on page 10-27
- “Align Text” on page 10-28
- “Link” on page 10-28
- “Unlink Text” on page 10-29
- “Convert Annotation” on page 10-29

Rotate

To rotate text to align with another entity, do the following:

1. Select the text.
2. Click **Modify** ▶ **Text** ▶ **Rotate** (Figure 10-29).
3. Click on a start point in the display area to change the pointer to an arrow or enter values into the input bar.
4. Move the arrow at the angle in which you want to rotate the text, and then click.

The text automatically rotates to the angle you selected.

5. Press **Esc** to end the routine.

Align Text

Use the Align command to align labels along existing text, arcs, lines, or two points (Figure 10-30).

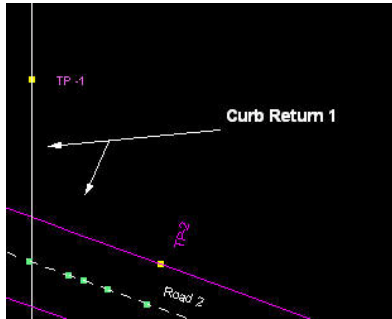


Figure 10-30. Aligned Text

To use the Align Text command, do the following:

1. Click **Modify** ▶ **Text** ▶ **Align** ▶ **Align Left/Align Right/Align Top/Align Bottom**.
2. Select an arc, line, text, or two points to align with, as stated in the prompt line in the lower left corner of the window.
3. Select the text you want to align.

The text moves according to the command you selected in Step 1 above.

Link

To link text to an entity, do the following:

1. Click **Modify** ▶ **Text** ▶ **Link Text**.
2. Select a point, line, arc, or alignment to link to, as stated in the prompt line on the lower left side of the window.
3. Select the text you want to link.
4. Press **Esc** to end the routine.

See also “Unlink Text” below.

Unlink Text

To unlink text to an entity, do the following:

1. Click **Modify ▶ Text ▶ Unlink Text**.
2. Select a point, line, arc, or alignment to unlink, as stated in the prompt line on the lower left side of the dialog box.
3. Click on (highlight) the text that it is linked to.
4. Press **Esc** to end the routine.

See also “Link” on the previous page.

Convert Annotation

Use the Convert Annotation command to convert the annotation for points, lines, arcs, or lots to ordinary text on display.

To use the Convert Annotation command, do the following:

1. Select currently annotated entities.
2. Click **Modify ▶ Text ▶ Convert Annotation**.

The *Annotation Conversion* dialog box appears (Figure 10-31).

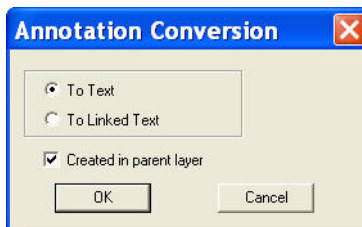


Figure 10-31. Annotation Conversion Dialog Box

3. Select one of the following:
 - **To Text** – converts the annotation to text. The text will remain if the entity is deleted at a later stage.
 - **To Linked Text** – converts the annotation to linked text. This type of text is linked to the entity and will be removed if the entity is deleted at a later stage.
4. Select the **Created in parent layer** check box to create the text in the same layer as the entity.
5. Click **OK** to convert the selected annotation to text.

The text style used for the annotation is also used for the text or linked text.

Extend/Trim Selection to Boundary

Use the Extend/Trim Selection to Boundary command to trim a line, arc, polyline, or string.

To extend or trim a selection to a boundary, do the following:

1. Click **Modify ▶ Extend/Trim Selection to Boundary**.
2. Follow the prompt, “*Select a line, arc or string*” at the bottom of the screen.
3. Follow the next prompt that says: “*Select boundary from list or pick boundary from screen, then press Trim*”.

NOTE: The boundary can be selected from the screen using the left-mouse button.

4. Select or enter a boundary name on the **boundary selection input bar** above and press **Trim** (Figure 10-32 on page 10-31). The *Extend/Trim Options* dialog box appears (Figure 10-32 on page 10-31).

Select the following options:

- Extend to Boundary
- Trim – check mark this box to enable a trim option:
 - Trim Outside: enable to trim the segments of the selected objects on the outside of the boundary, thus retaining the segments of the selected entities within the boundary.
 - Trim Inside: enable to trim the segments of the selected objects on the inside of the boundary, thus retaining the segments of the selected entities within the boundary.
 - Segment at Intersection: enable to break the entity at the intersection of the entity with the boundary.

A new point will be created at this intersection point.

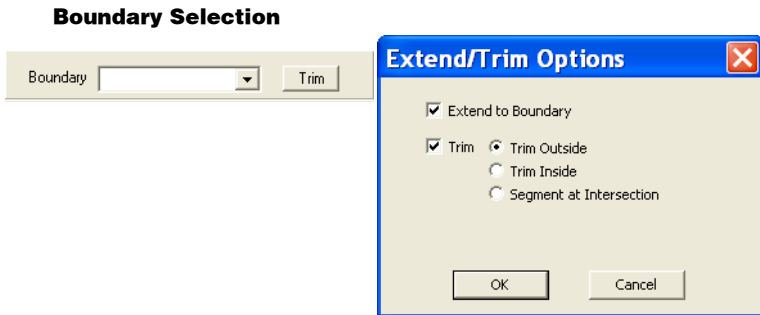


Figure 10-32. Extend or Trim a Selection to a Boundary



Multiple objects should be selected before attempting to trim. To quickly do this, click Ctrl+A (select all), then specify the boundary. All entities will be trimmed to the specified boundary using this method.

Properties

Use the Properties command to edit the attributes and properties of an entity. One entity can be selected and edited as a single item or a group of entities can be selected and edited as a group.

To display the properties of an entity, double-click the entity or select the entity and click **Modify ▶ Properties**. The properties dialog box for that entity appears.

For more information about properties of specific entities, see “Basic Data Types or Entities” on page 4-1.

Match Properties

Use the Match Properties command to duplicate the properties of an entity and apply them to another entity.

To Match Properties for an entity, do the following:

1. Click **Modify ▶ Match Properties**.

2. Select an entity from which to copy the properties, as stated in the prompt line in the lower-left corner of the window.
3. Select an entity to which you want to apply the properties you copied. You can select multiple entities.
4. Press **Esc** to end the routine.

Create Points from Entities

This option creates point on entities without points. The new points will be placed in the same layer as the entity and use the current settings to set it's properties.

Delete Points from Entities

This option deletes points from selected entities. The entities will be retained but the points will be deleted from the database.



Use this command to delete existing points and their properties. The properties of the existing points cannot be recovered. Use the Undo command if the command is run by accident.

Explode

Use the Explode String command to convert a string or polyline back into points and individual lines:

1. Click **Modify ▶ Explode**.
2. Click on a string or polygon.

The entity is removed and replaced by points joined by lines.



If it is difficult to select a polygon, then select it first and then click the **Explode** command.



Boundaries are a form of polygon. If a boundary is selected, then a confirmation message may appear.



Polylines do not contain points. When exploding a polyline the result will be line segments without points. To create points on these segments use the Create Points from Entities option found in the modify menu.

Filter Points

Use the Filter Points command to reduce the number of points contained in a nominated string.

1. Click **Modify ▶ Filter Points**.

The *Filter Tolerance* dialog box appears (Figure 10-33).

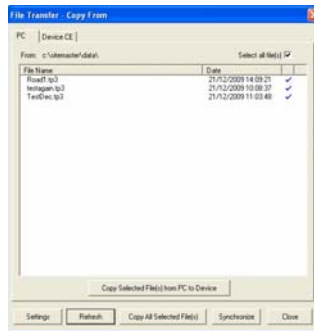


Figure 10-33. Reduce Points in a Nominated String

2. Do the following to delete or change the points within the filter (Figure 10-34 on page 10-34):
 - Horizontal tolerance – enter a value based on the position of points before and after each string point.

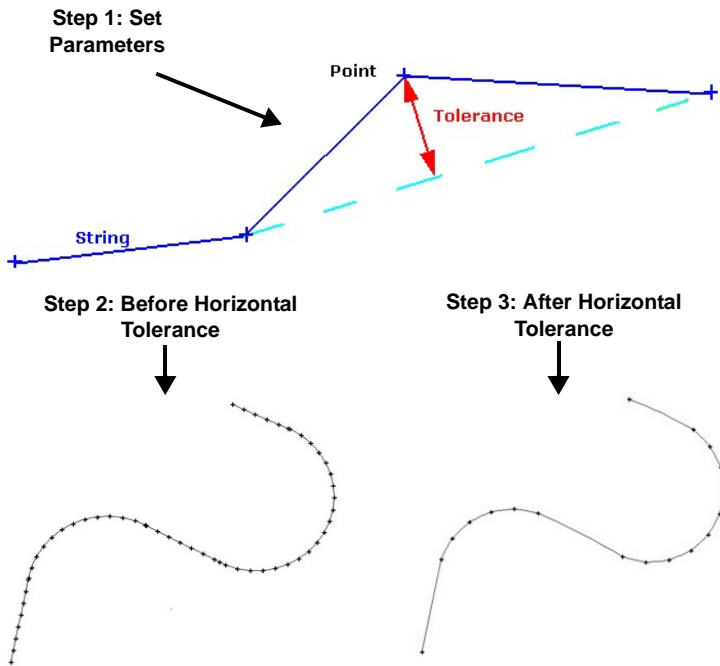
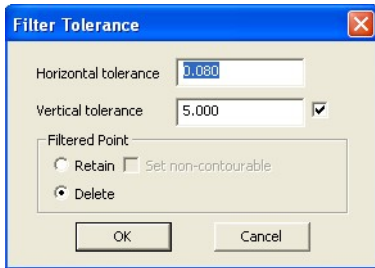
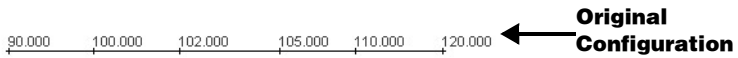


Figure 10-34. Filter Points

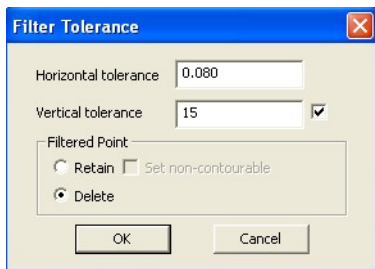
- Vertical tolerance – enable this option to delete collinear points in 3D Strings.

When vertical filtering is not applied, points within the specified horizontal tolerance are deleted regardless of the elevation. When vertical filtering is applied, only points within both the horizontal and vertical tolerances are deleted. Figure 10-35 and Figure 10-36 on page 10-35 show examples of vertical filtering.



Results after
5' vertical
filter setting

Figure 10-35. Vertical Filtering – After 5' Vertical Filter Setting



Results after
15' vertical
filter setting

Figure 10-36. Vertical Filtering – After 15' Vertical Filter Setting

Copy

Do the following to Copy an entity (Figure 10-37 on page 10-36):

1. Click **Modify** ► **Copy**.
2. Select the first source point, as stated by the prompt line on the lower left side of the window.
3. Select the second source and destination point, as prompted

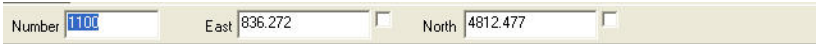


Figure 10-37. Copy an Entity

Move

Use the Move command to move a single entity to another position within the same MAGNET Office project.

1. Select the entity by clicking on it or dragging a window around it.
2. Click **Modify ▶ Move**.
3. Follow the directions on the prompt line on the lower left side of the dialog box.

The *Move Confirmation* dialog box appears (Figure 10-38).

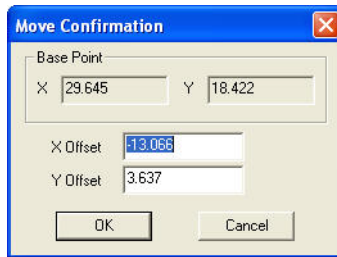


Figure 10-38. Move Confirmation Window

4. Make sure the coordinates are accurate, and then click **OK**.
The entity is positioned at the specified coordinates.

OR:

1. Select the entity by clicking on it or dragging a window around it.
2. Press and hold the **Ctrl** key.
3. Move the pointer over the entity's center point or handle.

The pointer changes to the **Move Entity** \leftrightarrow pointer

4. Click and drag the entity to a new position.

See also “Preventing Moving Entities” on page 3-3.

Rotate

Use the Rotate command to rotate a single entity to another position within the same MAGNET Office project:

1. Select the entity by clicking on it or dragging a window around it.
2. Click **Modify ▶ Rotate**.
3. Follow the directions on the prompt line on the lower left side of the window.

The *Rotate Confirmation* dialog box appears (Figure 10-39).

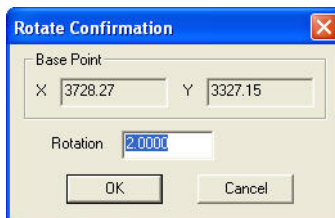


Figure 10-39. Rotate Confirmation Window

4. Make sure the coordinates are accurate, and then click **OK**.
The entity is positioned at the specified coordinates.

Align

The Align command allows 2D alignment of an object by specifying source and destination points.

To use the Align command, do the following:

1. Select one or more objects to align.
2. Click **Modify ▶ Align**.
3. Select the first source point, as stated by the prompt line on the lower left side of the window.
4. Select the first destination point.

5. Select the second source and destination point, as prompted. A confirmation message displays (Figure 10-40).

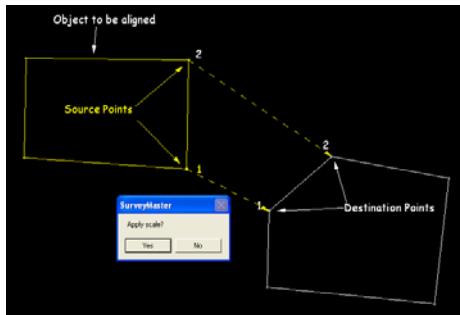


Figure 10-40. Scale Message

6. On the message, press either **Yes** or **No**.
 - **Yes** – matches the source points exactly with the destination points (Figure 10-41).

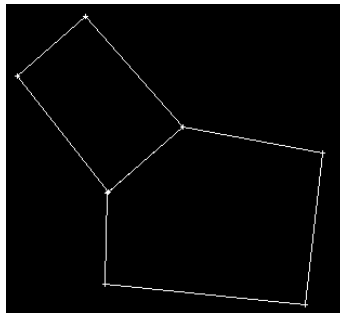


Figure 10-41. Results – Yes

- **No** – matches the first source and destination point exactly and any difference will be shown in the second source and destination points. This is useful when comparing calculated deed points with field surveyed points (Figure 10-42 on page 10-39).

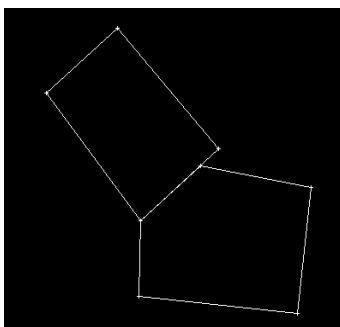


Figure 10-42. Results – No

Align existing DTMs You can now align existing DTMs. Do the following to align DTMs:

1. Click **Modify ▶ Align**.
2. Select an object.
3. Left-click on the mouse to display the *Align DTM* dialog box (Figure 10-43).
4. Enable either *Current DTM*, *All DTM*, or *Selected DTM* to align a DTM.
5. Follow steps 3 through 6 as outlined above.

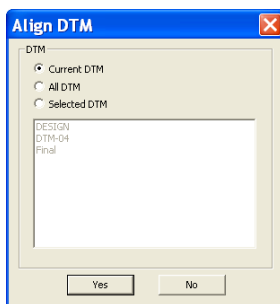


Figure 10-43. Align a DTM



To align objects without points, be sure to select the objects **BEFORE** executing the command.

Scale

Use the Scale command to change the size of a single entity within the MAGNET Office project:

1. Select the entity by clicking on it or dragging a window around it.
2. Click **Modify ► Scale**.
3. Select the base point, as stated in the prompt line in the lower left corner of the window.
4. Enter the scale in the input bar, and then press **Enter**.

The *Scale Confirmation* dialog box appears (Figure 10-44).

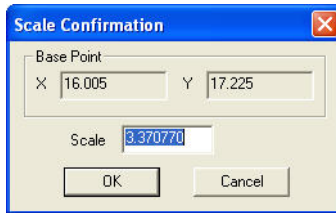


Figure 10-44. Scale Confirmation Window

5. Make sure the scale is accurate, and then click **OK**.
The entity is resized at the specified scale.

Offset

The Offset command offsets lines, arcs, or circles by specifying a right or left offset distance.

To use the offset command, do the following:

1. Click **Modify ► Offset**.
2. Select a line, arc, or circle, as stated in the prompt line in the lower-left corner of the dialog box.

The input bar appears (Figure 10-45).

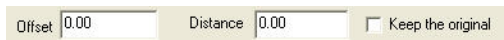


Figure 10-45. Input Bar

3. Enter an offset. A left offset is entered with a negative (-) prefix.

- Optionally, enter a **Distance** value. This allows for a positive or negative shift in the direction of the line. The direction of the line depends upon which end is picked when specifying the object for offset.
- Check mark the **Keep the original** box to retain the object to be offset.

Refer to the following example (Figure 10-46).

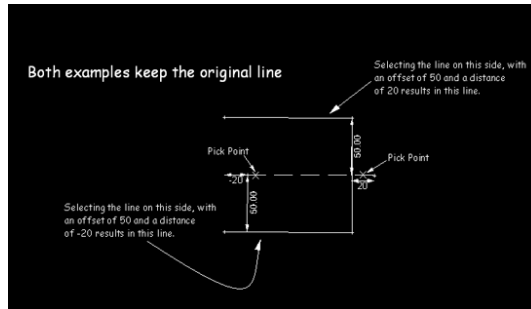


Figure 10-46. Offset Example

Survey View: Inquiry Menu

This section describes the different Survey View menu commands on the Inquiry menu located on the main menu bar (Figure 11-1).

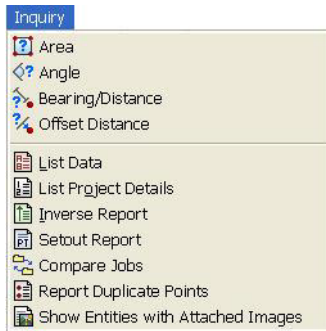


Figure 11-1. Survey View – Inquiry Menu

In This Section:


- “Area”
- “Angle” on page 11-3
- “Bearing/Distance” on page 11-4
- “List Data” on page 11-5
- “List Project Details” on page 11-6
- “Inverse Report” on page 11-7

Area

Use the Area command to calculate an area and perimeter of a closed figure that can be defined by selecting an existing polygon, existing

lines, or by temporarily creating a boundary by clicking points around the area.

To use the Area command, do the following:

1. Click **Inquiry** ► **Area** or click the *Query Area*  icon on the **Survey Tools** toolbar (Figure 11-2).

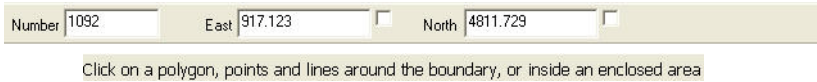


Figure 11-2. Calculation Area Prompt

2. Follow the prompts for polygon, points, and lines around the boundary:
 - An edge line of a polygon – When an existing polygon is selected, the *Area Calculation* dialog box appears (Figure 11-3 on page 11-3).
 - A point which will define one corner of the area boundary – Points can be selected by pointer, point number, or X and Y coordinate value. Select the first point and additional points until the area closes back to the first selected point, and then select **Calculate Area** from the right-click shortcut menu.
 - A line which will be one leg of the area boundary – When a line, arc, or string is selected, a path is followed along the lines until a closed boundary area is automatically selected and the *Area Calculation* dialog box appears (Figure 11-3 on page 11-3).

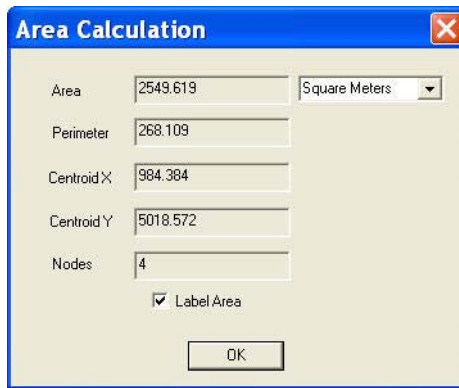


Figure 11-3. Area Calculation Window

3. Parameters for the area display.
4. Select either *Square Meters* or *Hectares*, then click **OK**.

Angle

Use the Angle command to calculate the angle between two lines, defined by three points, including the point where the two lines join (Figure 11-4).

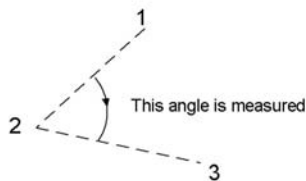


Figure 11-4. Calculate Angle

To use the Angle command, do the following:

1. Click **Inquiry** ▶ **Angle**.
2. Click on a start point, line, or string line segment. Or you can enter a point name in the input bar, and press **Enter** (Figure 11-5).



Figure 11-5. Point Name Input Bar

3. Click another point, line, or string line segment where the two lines join or intersect.

The *Inquire Angle* dialog box appears (Figure 11-6), displaying the computed angles and the three coordinate points.

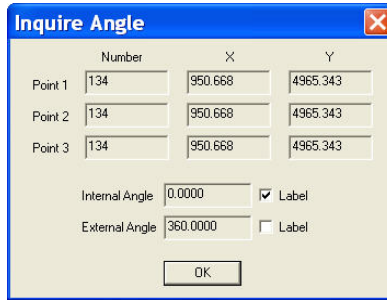


Figure 11-6. Enquire Angle

4. Click **OK** to accept the calculation.

Bearing/Distance

Use the Bearing Distance command to calculate the bearing and distance between any two points, or the bearing and distance of an existing line. When the selected points have elevations, the change in height and slope details are also displayed.

To use the Bearing Distance command, do the following:


1. Click **Inquiry** ► **Bearing/Distance** or click the *Enquire Bearing and Distance*  icon on the **Survey Tools** toolbar.
2. Select a line or the start point or type the two point numbers on input bar (Figure 11-7).



Figure 11-7. Bearing/Distance Input Bar

3. Select the second point.

The *Inquire bearing and distance* dialog box appears, displaying details of the two points and their relative positions (Figure 11-8).

	Eastings	Northings	Elevation
From	17	-155.954	627.232
To	18	260.074	344.032
			0.000
			0.000
Bearing	Distance		
124.1438	503.271		
Change In Height	Slope Distance	Slope	Slope %
0.000	0.000	Zero Slope	0.000

Figure 11-8. Inquire Bearing and Distance

4. Click **OK** to save.

List Data

Use the List Data command to list the data of a selected entity. This command has various filters to assist with data selection. Unless all active data in a project is required on the list, use this command to filter the list.

To use the List Data command, do the following:

1. Click **Inquiry ▶ List Data**.

The *List Data Selection* dialog box appears (Figure 11-9 on page 11-6).

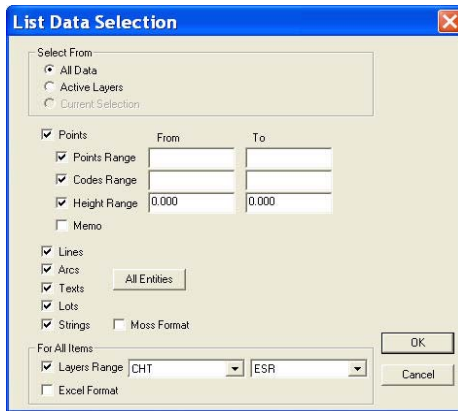


Figure 11-9. List Data Selection

2. Choose **All Data** to include all the data currently displayed in the Survey View in the report or choose **Current Selection** to include data for the currently selected entity.
3. Check mark the following parameters for inclusion in your report: **Points**, **Lines**, **Arcs**, **Text**, **Lots**, and **Strings**.
4. To filter the selection of points, check mark the **Points Range**, **Codes Range**, and/or **Height Range** boxes and enter the appropriate values.
5. To filter the selection by layers, check mark the **Layers Range** box and select the range of layers using the drop-down menus.
6. When you have finished, click **OK** to close the dialog box and display the data listing.

List Project Details

Use the List Project Details command to display the job information details when creating the project. An example of the report, which can be saved as a (.rep) file or printed, is shown in Figure 11-10 on page 11-7.

To list project details, do the following:

1. Click **Inquiry** ► **List Project Details**.

The *List Job Details* dialog box appears (Figure 11-10).



Figure 11-10. List Job Details

2. Choose whether to list the details of all the projects in the database or just the current project, and then click **OK**. A report appears, displaying all of the project details for the current project or all projects, depending on your selection (Figure 11-11).

```

Path: E:\masters-image-*****\Data
JobName      FileSize      Created        Modified       Time
ROADXS**    ***.* KB      **/**/****    **/**/****    **:**:**
Company      : Company     :
Name         : Name        :
Comment      :
Details*    :
Details*    :
Details*    :
Details*    :
Title       :

```

Figure 11-11. Project Details Report

Inverse Report

To use the Inverse Report command:

1. Click **Inquiry ▶ Inverse Report**.

The *Inverse Report Options* dialog box appears providing three inverse options:

- Radiations – a sideshot inverse. One point is occupied and all the inverses come from this point to the specified points.
- Traverse – moves the occupied point as the inverses are specified. Inversing around a property description is an example of a traverse inverse.

- Two Points – inverses between specified pairs of points (Figure 11-12).



Figure 11-12. Inverse Report Options

There are two modes for inverse reports:

- Bearing – If the angle display settings are set to Bearing (Quadrant) in the project settings, then all inverses display in bearing quadrant values. For example, for N 25°15’34” E, the inverse report will dock at the bottom of the screen and will remain open until the **Close** option is selected.
- Angle – If you are using an angle, then three points will need to be specified. For example, entering 3, 4, 5 results in the angle being calculated between lines 4-3 and 4-5. If the angle display settings are set to Azimuth (WBC) in the project settings, then all inverses display in azimuths when selecting the Bearing (Quadrant) option.

An example of a sideshot inverse using bearing (quadrant) is shown in Figure 11-13.

To	Bearing	Distance	Northing	Easting	Height	Code
55	S89°45'22"E	111.48	-1258.90	1065.29		PT
65	S47°55'48"E	149.95	-1358.90	1065.12		PT
64	S0°05'51"W	100.00	-1358.43	953.63		PT
46	S86°19'59"W	356.72	-1281.24	597.81		PT
45	S64°30'15"W	414.23	-1436.73	579.92		PT

Figure 11-13. Example of a Sideshot Inverse

2. Make the necessary selections, and then click **OK**.

The Inverse Report appears under the display area (Figure 11-14).



Figure 11-14. Example of an Inverse Report

3. Depending on the options you selected, you may be required to enter a point name in the input bar or click on a point in the display area.
The Inverse Report updates with every point name that is entered or selected.
4. When you have finished selecting points, press **Esc** to end the routine.
5. To save, edit, print, or close the inverse report, click the appropriate button on the lower right side of the window (Figure 11-14).

Notes:

Survey View: Survey Menu

This section describes the different Survey View menu commands on the Survey menu located on the main menu bar (Figure 12-1).

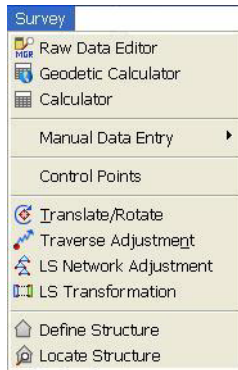


Figure 12-1. Survey Menu in Survey View

In This Section:

- “Raw Data Editor” on page 12-2
- “Geodetic Calculator” on page 12-2
- “Display the Calculator” on page 12-5
- “Manual Data Entry” on page 12-5
- “Control Points” on page 12-21
- “Translate/Rotate Data” on page 12-22
- “Traverse Adjustment” on page 12-26
- “LS (Least Squares) Network Adjustment” on page 12-28
- “Using the Data Editor Observations and Points Tabs” on page 12-33

- “Define Structure” on page 12-44
- “Locate Structure” on page 12-45

Raw Data Editor

Use the Raw Data Editor command (Figure 12-2) to download and/or import raw files from supported data recorders. These files automatically get translated into a “neutral” file format. To use this command, click **Survey ▶ Raw Data Editor**. Select a project name from the Raw Data Manager, and click **OK** to open the Raw Data Editor.

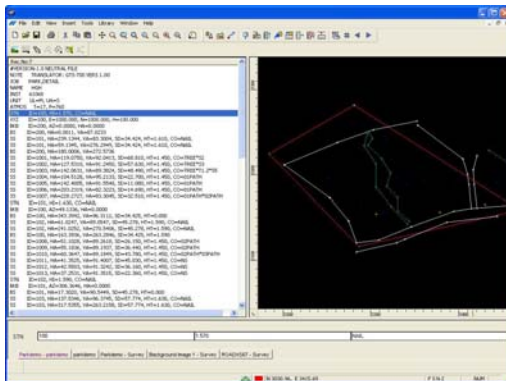


Figure 12-2. Raw Data Editor

Geodetic Calculator

Use the Geodetic Calculator command to calculate map projection (grid) coordinates using geodetic coordinates and vice versa. MAGNET Office calculates coordinates either directly using known geodetic coordinates or inversely using a known grid system.

If you are applying a geoid model to the elevation computations, a geoid file (*.gff* or *.jff*) must be available. A geoid file provides information about the separation between the purely geometric, ellipsoidal representation of the earth and the physical model of the earth that closely approximates mean sea level (the geoid). For

example, use a geoid model to obtain approximate mean sea level heights from GPS measured ellipsoidal heights.

Direct Coordinate Calculation To calculate grid coordinates based on given geodetic coordinates:

1. Click **Survey ▶ Geodetic Calculator**.

The *Coordinate Calculator* dialog box appears (Figure 12-3).

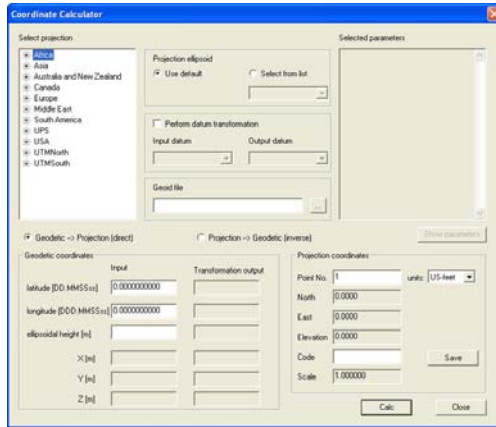


Figure 12-3. Coordinate Calculator

2. In the **Select projection** window, choose the projection and geoid (optional) to use in the calculation.
3. Click **Geodetic ▶ Projection (direct)** to compute the grid coordinates.
4. Enter either WGS or local geodetic coordinate values in the **Geodetic coordinates** window in the **Input** fields.
 - Use negative values for West longitude and South latitude.
 - Enter the ellipsoidal height in the same unit currently set for distances in the project.
5. Click **Calc** to calculate the corresponding geodetic coordinates (WGS or local datum) and the grid coordinates based on the selected projection.
6. Click **Save** to save the calculated grid coordinates to the project point database.

Inverse Coordinate Calculation To calculate geodetic coordinates based on given grid coordinates:

1. Click **Survey** ► **Geodetic Calculator**.

The *Coordinate Calculator* dialog box appears (Figure 12-4).

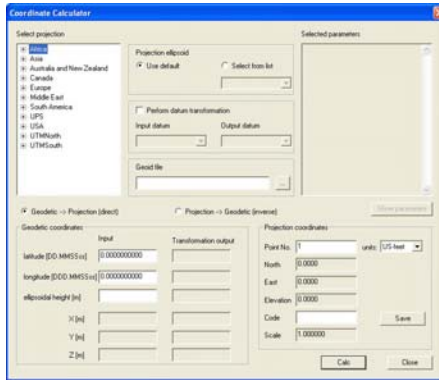


Figure 12-4. Coordinate Calculator

2. In the *Select projection* dialog box, choose the projection and geoid (optional) to use in the calculation.
3. If the coordinates are in a known grid system, click **Projection** ► **Geodetic (inverse)** to compute the grid coordinates.
4. Enter coordinate values in the **Projection coordinates** dialog box.

If a geoid file has been specified, MAGNET Office will enter the geoid height as determined from the file.

5. Click **Calc**.
MAGNET Office calculates the corresponding geodetic coordinates based on the selected projection.
6. Click **Save** to save the calculated geodetic coordinates to the project point database.

Display the Calculator

Use the Calculator command to open the Microsoft Windows® calculator.

Manual Data Entry

The Manual Data Entry menu has options for keyboard data entry typed from field book records (Figure 12-5).

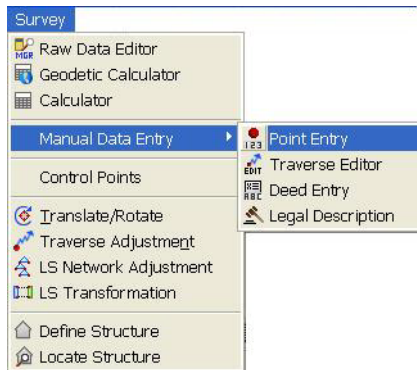


Figure 12-5. Manual Data ENTRY Sub-Menus

In This Section:

- “Point Entry”
- “Traverse Editor” on page 12-8
- “Deed Entry” on page 12-13
- “Legal Description” on page 12-17

Point Entry

Use the Point Entry command to manually enter or edit a series of points of known coordinates and elevations, and save the points into the project database.

To manually enter or edit a series of points of known coordinates and elevations, do the following:

1. Click **Survey** ► **Manual Data Entry** ► **Point Entry**.
The *point coordinate entry* dialog box appears (Figure 12-6).

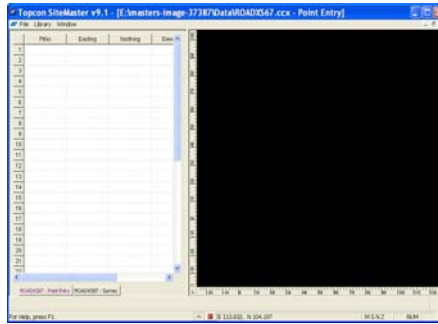


Figure 12-6. Point Coordinate Entry Window

The left side of the editor is for data entry and the right side is for a graphical display of data as it is entered. You can configure the left side for the data type and the order in which the data is entered.

2. To configure the editor, right-click and select **Configure** from the shortcut menu.

The *Configuration* dialog box appears (Figure 12-7).

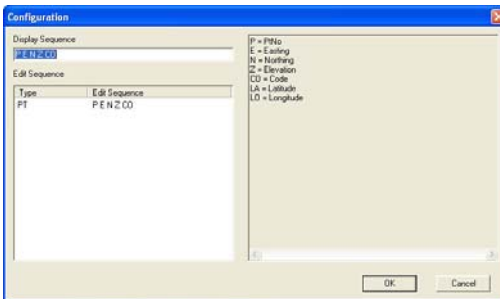


Figure 12-7. Configuration Dialog Box

3. To set the data type and display sequence order, click in the **Display Sequence** field, and using the codes for the available data shown on the right side, enter the data in the desired order you want it displayed, separated by a space.
4. To define an edit sequence, click on the **Edit Sequence** field and enter the order for editing a record. The edit sequence can differ from the display sequence. You may want to set the edit sequence

to where it only includes the bearing and distance and not the point number and code for quick editing.

5. Click **OK** to finish.
6. To set the content and data order right-click and select **Insert Column** from the shortcut menu. This enables you to insert a data column, and drag and drop the displayed columns into the desired order.



The active column is listed in the hide column option and only the columns that aren't actively displayed are available for insertion.

7. In the *Point Coordinate Entry* dialog box (Figure 12-8 on page 12-8), enter the coordinates, height, and point code for each point. If the point field is left empty, MAGNET Office automatically allocates the next available point number when the data is saved. For column descriptions, see Table 12-1 on page 12-8.

If a code from the Survey Code Library is used, then the features associated with the code will be applied as properties to the point, and the point will be saved on the named layer.

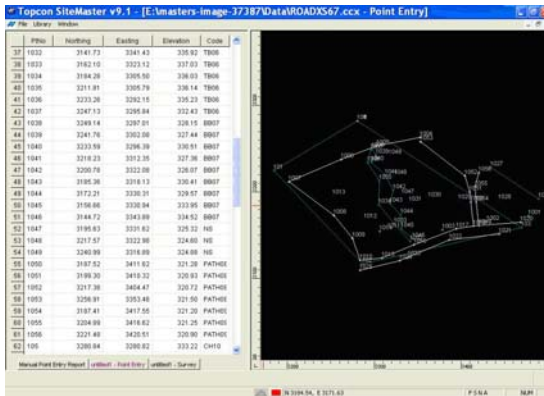


Figure 12-8. Coordinated Points Entry

Table 12-1. Point Coordinate Entry Column Descriptions

Column Name	Column Description
Pt No (P)	Enter a point number or leave the field empty for automatic allocation of the next available point number.
Easting (E)	Enter the known Easting coordinate for the point.
Northing (N)	Enter the known Northing coordinate for the point.
Elevation (Z)	Enter the known height for the point.
Pt Code (CO)	Enter the point code. This code may be a feature code from the Survey Code Library. The code is carried forward from one row to the next during reduction, so only enter a code when it changes.
Latitude (LA)	Enter the known point latitude. A coordinate system must be selected in the project settings for entry of latitude and longitude.
Longitude (LO)	Enter the known point longitude. A coordinate system must be selected in the project settings for entry of latitude and longitude.

Traverse Editor

Use the Traverse Editor command to input traverse data for reduction and adjustment using the Bowditch method or transit rule. The traverse can be entered as observed angles, quadrant bearings, bearings in units of degrees, minutes and seconds, or decimal degrees

or grads. Distances may be entered as slope or horizontal in meters, feet, decimal feet, or links.



The Traverse Editor format displayed on screen depends on the local requirements and the license configuration in use.

To use the Traverse Editor command, do the following:

1. Click **Survey ▶ Manual Data Entry ▶ Traverse Editor**.
The *Traverse Data Manager* dialog box appears (Figure 12-9).

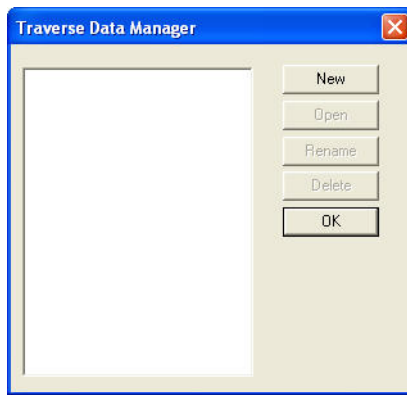


Figure 12-9. Traverse Data Manager

2. On the *Traverse Data Manager* dialog box, you can do the following:
 - **New** – creates a new traverse
 - **Open** – opens a previously saved traverse
 - **Rename** – changes the name of a traverse
 - **Delete** – deletes a traverse
 - **OK** – saves the routine

Create a Traverse To create a new traverse, click **New** on the *Traverse Data Manager*. A *Configuration* dialog box appears (Figure 12-10 on page 12-10).

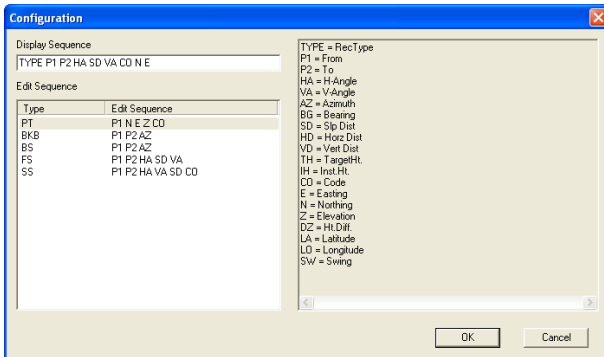


Figure 12-10. Data Entry Dialog Box

Table 12-2 describes the data entry column fields.

Table 12-2. Data Entry Column Descriptions

Column Name	Column Descriptions
TYPE= RecType	Record type, example: back sight, foresight, side shot, etc.
P1=PtFrom	Enter a point number for the station number that the instrument is set upon. This is the start station of the traverse. This point should exist with known coordinates in the Survey View.
P2=PtTo	Enter a point number for the station number that is being observed from the instrument. This usually becomes the next From point in the traverse.
HA= H-Angle	Enter the horizontal angle for the set up station to the observed point in this column.
VA= V-Angle	Enter the vertical angle from the set up station to the observed point in this column. Leave this field blank or enter a horizontal vertical angle if horizontal distances are being entered.
AZ= Azimuth	Enter the azimuth angle from the set up station to the observed point in this column.
SlopeDist.	Enter the slope distance from the set up station to the observed point in this column. If horizontal distances are being entered, this will be the horizontal distance if the vertical angle field is left empty.
Inst.Ht.	Enter the instrument height at the set up point. The height is carried forward from one row to the next during reduction, so only changed heights need to be entered.

Table 12-2. Data Entry Column Descriptions

Column Name	Column Descriptions
TargetHt.	Enter the height of the target at the observed point. The target height is carried forward from one row to the next during reduction, so only changed target heights need to be entered.
Easting	Enter the known Easting coordinate for the point.
Northing	Enter the known Northing coordinate for the point.
Elevation	Enter the known height for the point.
Code	Enter the code for the observed point. This code may be a feature code form the Survey Code Library. String numbers and comments may also be used. The code is carried forward from one row to the next during data reduction, so codes only need to be entered when they change.

Traverses can be defined as:

- A loop, where the traverse closes on the start station. The start and end stations are the same point with known coordinates.
- Open ended, where the start and end stations are different, but the coordinates of both points are known.

Traverses may also include side shots or radiations. These are calculated from the closed and adjusted traverse points.

All points in a traverse may have computed heights (Figure 12-11).

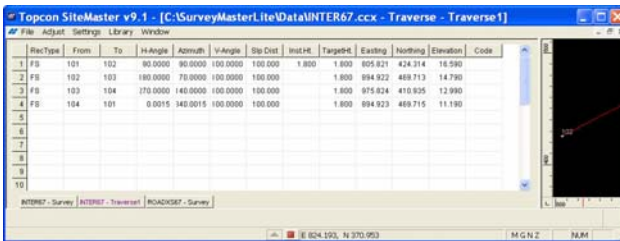


Figure 12-11. Data Entry Window – Traverse

The traverse starts on station point 101 and passes through stations 102, 103, and 104 to close back on point 101.

Traverse Adjustment To adjust the traverse, do the following:

1. Click **Traverse ▶ Adjust**. The *Control Point Coordinate* dialog box appears (Figure 12-12).

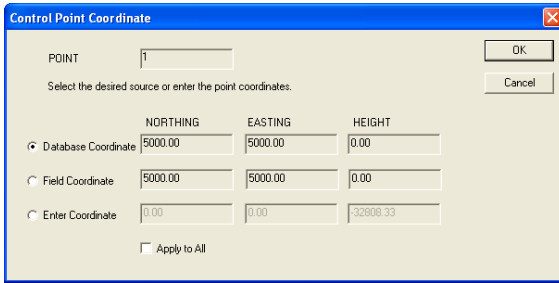


Figure 12-12. Control Point Coordinate

2. Select a source or enter coordinates, and then click **OK**. The *Traverse Adjustment* dialog box appears (Figure 12-13).

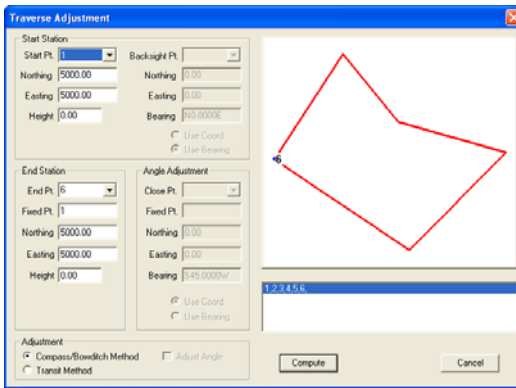


Figure 12-13. Traverse Adjustment Dialog Box

- Modify the properties as needed, and then click **Compute** to display the **Unadjusted Traverse** dialog box (Figure 12-14).

	ID	East	North	Elevation
Fixed Pt	1	5000.00	5000.00	0.00
Observed Pt	6	5000.06	4999.96	0.00
		East Diff	North Diff	Elev Diff
Coordinate Misclose		-0.06	0.04	-0.00
Misclose Bearing		N59.1059	Distance	0.08
Angular Error		0.0000	Per Station	0.0000
Enclosed Area		5.67	Accuracy	1:29021
Traverse Length		2181.14	No of Setup	5

Adjust Elevations

Click OK to reduce side shots and save the points.

Buttons: Angles, Coordinates, OK, Cancel

Figure 12-14. Unadjusted Traverse

- Click **Coordinates** to adjust the coordinates, or click **OK**.

Deed Entry

Use the Deed Entry command to enter line and curve data that comprise deed descriptions. With this command, you can:

- Customize the data type and order displayed
- Add or hide data columns using the right-click shortcut menu
- Save deeds for future reference and drawing
- Edit the order of the data entry by dragging the columns to place them in the required order

In addition, there is an optional automatic graphic display of points, lines, and curve data as it is entered.

To create a deed entry, do the following:

- Click **Survey** ► **Manual Data Entry** ► **Deed Entry**.
The *Deed Entry Manager* dialog box appears (Figure 12-15 on page 12-14).



Figure 12-15. Deed Entry Manager

2. On the *Deed Entry Manager* dialog box, you can do the following:
- **New** – creates a new deed.
 - **Open** – edits or views an existing deed.
 - **Rename** – changes the name on an existing deed.
 - **Delete** – removes an existing deed.
 - **OK** – accepts any changes and ends the command.

New Selecting **New** opens the *deed entry/editor* dialog box (Figure 12-16).

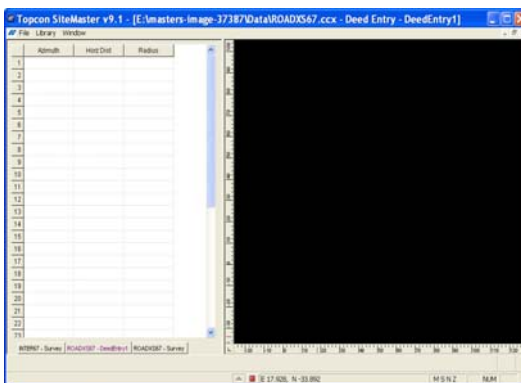


Figure 12-16. Deed Editor

The left side of the editor is for data entry, and the right side shows a graphical display of data as it is entered. You can configure the left side for the data type and the order in which the data is entered.

1. To configure the editor, right-click and select **Configure** from the shortcut menu.

The *Configuration* dialog box appears (Figure 12-17).

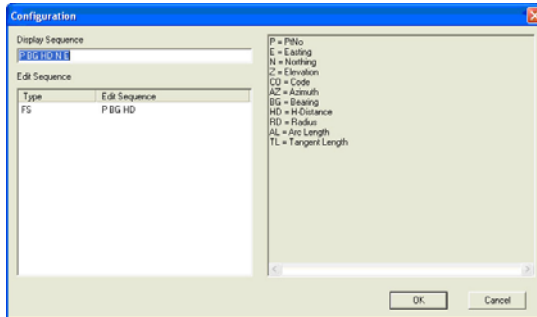



Figure 12-17. Deed Editor – Configuration

2. To set the data type and display sequence order, click in the **Display Sequence** field, and using the codes for the available data shown on the right side, enter the data in the desired order you want it displayed, separated by a space.
3. To define an edit sequence, click on the **Edit Sequence** field and enter the order for editing a record. The edit sequence can differ from the display sequence. You may want to set the edit sequence to where it only includes the bearing and distance and not the point number and code for quick editing.
4. When you are finished, click **OK**.
5. You can also set the content and data order by right-clicking and selecting **Insert Column** from the shortcut menu. This enables you to insert a data column, and drag and drop the displayed columns into the desired order.



The active column is listed in the hide column option and only the columns that aren't actively displayed are available for insertion.

- To display points, lines, and arcs on the graphic preview, click the **Hot Button**  to open the *Display Settings* dialog box (Figure 12-18).

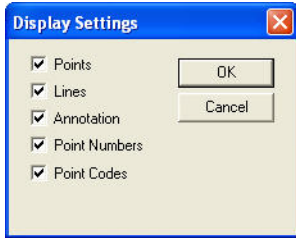


Figure 12-18. Display Settings Window

- Select the options you want to display, and click **OK**.
- In the *Deed Entry* dialog box, enter the coordinates and point number for each point (Figure 12-19).

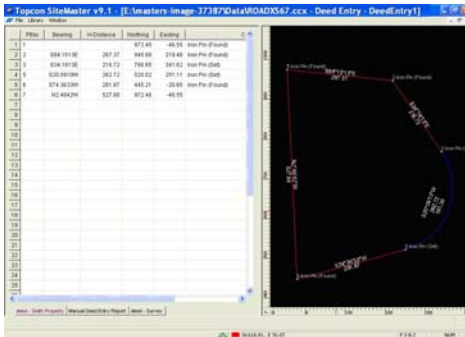


Figure 12-19. Example Deed Entry

- To generate a traverse closure report, click **File ▶ Report** (Figure 12-20 on page 12-17).

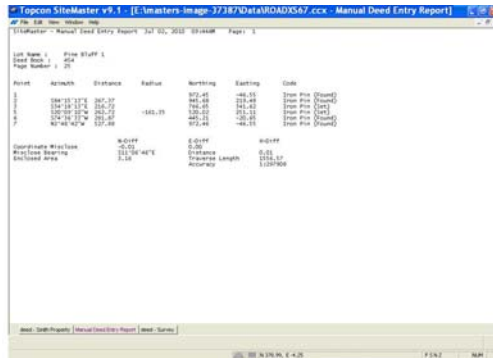


Figure 12-20. Coordinated Points Entry

Table 12-3 describes File menu options on the traverse closure report.

Table 12-3. File Menu Descriptions

Field Name	Field Description
Save Deed	Saves deed to database.
Save to Survey	Draws the lines, arcs, and points in the Survey View. It also saves the points to the database.
Report	Generates a traverse closure report. A report can be generated at anytime during data entry by selecting this option.
Header Details	Places optional information at the top of the deed report.
Survey View	Switches to Survey View.
Close Deed Editor	Ends command.

Legal Description

Use the Legal Description command to create a legal description of the land parcel and save these descriptions to a file and/or directly to the drawing. Use the Legal Description Template design, located in the library, to define the components to include in the legal description. The available components include the following:

- Title
- Opening Description
- Commencement Begin Description

- Commencement End Description
- Reference to Description
- Boundary Begin Description
- Boundary End Description
- Closing Description
- Line Description
- Arc Description
- Header and Footer

To create a legal description of land parcels, do the following:

1. Click **Survey ▶ Legal Description**.

A prompt appears requesting you to click on an enclosed area.

2. Click to select any line that forms an enclosed area or click on an enclosed area made up of lines or strings. A confirmation message appears (Figure 12-21).

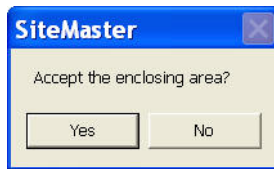


Figure 12-21. Confirmation Window

3. Click **Yes** to accept the enclosed area.

MAGNET Office assumes a starting point indicated by a solid red circle.

- Press **Esc** to accept this start point, or click to select your own start point and then press **Esc** (Figure 12-22).

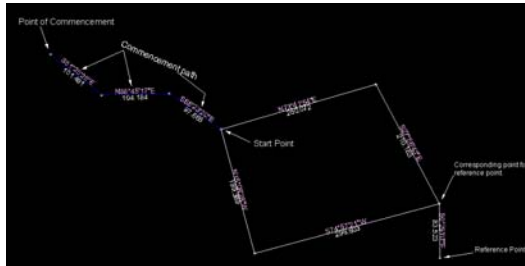


Figure 12-22. Start Point and Commencement Path

- Select a point, line, or string (the commencement path), and then press **Esc**.

The commencement path is comprised of more than one course beginning at a reference point and leading to the beginning point of the legal description.

- Click to select a reference point, and then press **Esc**.

The *Legal Description Settings* dialog box appears (Figure 12-23).

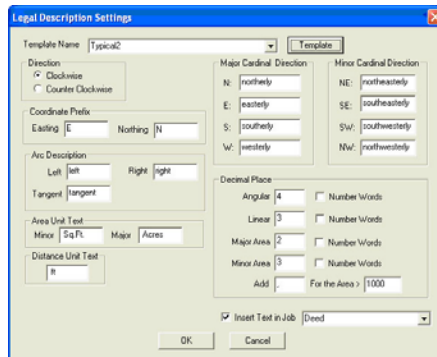


Figure 12-23. Legal Description Settings

- Modify the settings as required. For a description of the fields, see Table 12-4 on page 12-20.
- Click **OK** to accept the changes.

The property description displays in the internal editor and can be edited. Closing the property description editor, or switching to the

Survey View results in the description being displayed in the drawing. The description is automatically selected and is ready to be moved to the desired location within the drawing.

Table 12-4. Legal Description Settings Descriptions

Field Name	Field Description
Template Name	The name of the description template to be used in the legal description creation. The template can be selected from the library of description templates (if present) or newly created by selecting Template .
Direction	Determines the direction of the description.
Major Cardinal Direction	Determines the terminology for the major cardinal direction, when encountered, if specified in the description template.
Minor Cardinal Direction	Determines the terminology for the minor cardinal direction, when encountered, if specified in the description template.
Coordinate Prefix	Determine prefix for coordinate listings if specified in the description template.
Arc Description	Determines terminology for curve direction and tangent if specified in the description template.
Area Unit Text	Determines the suffix units to be used for minor and major area units if specified in the description template
Distance Unit Text	Determines the suffix unit to be used for distances in the description. The text value can be changed to a symbol, such as ' for feet.
Decimal Place	<p>These settings control the decimal place settings for angular, linear, and area values.</p> <p>Number words can be used in place of numerical digits for any of the measurement data. For example: the distance of 134.50 would be displayed as one hundred thirty-four point five zero.</p> <p>The Add field enables you to add a comma to area values greater than 1000, resulting in an area value of 1,000 when displayed.</p>
Insert Text in Job	If turned on, this option inserts the description text into the active project as multiline text.

Control Points

Control points are reference points at or near the project sight that provide access to the project coordinate system. Control files contain coordinate data for the control points. In the case of GPS applications, they also provide a mathematical link between the GPS and project coordinate systems.

To add, edit, delete, control, export or import control points, click **Survey ► Control Points**. The *Control Points* dialog box appears (Figure 12-24).

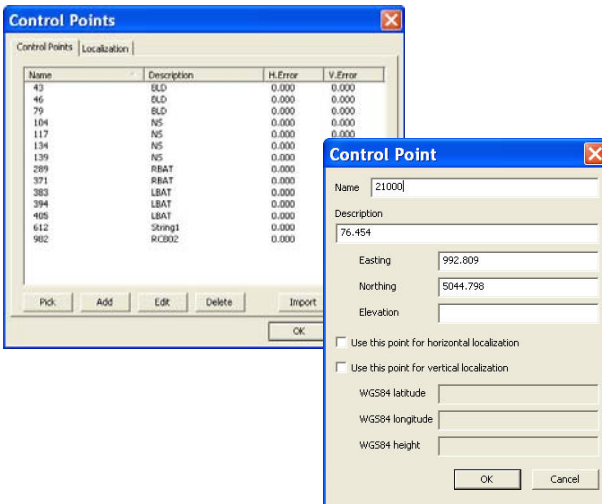


Figure 12-24. Control Points

Table 12-5 describes the field on the Control Points table.

Table 12-5. Control Point Field Descriptions

Field Name	Field Description
Name	Control point name.
Description	Control point description.
H.Error	Represents the probably horizontal error in the point position.
V.Error	Represents the probably vertical error in the point position
Pick	Press to pick a control point on the main screen.

Table 12-5. Control Point Field Descriptions (Continued)

Field Name	Field Description
Add	Press to add a control point on the Control Points table (Figure 12-24).
Edit	Press to edit the highlighted control point on the Control Points table (Figure 12-24).
Delete	Press to delete the highlighted control point on the Control Points table (Figure 12-24).
Import	Press to import a control point (Figure 12-24).
Export	Press to export a control point (Figure 12-24).
OK	Press to save (Figure 12-24).
Cancel	Press to quit the routine (Figure 12-24).

Translate/Rotate Data

Use the Translate/Rotate command to translate (move), rotate, or scale all of the data or selected data. Move the data to a new coordinate position, or rotate the data around a nominated point. You can also scale the data around a nominated point. DTMs may only be translated if no selection is made (Figure 12-25 and Figure 12-26).

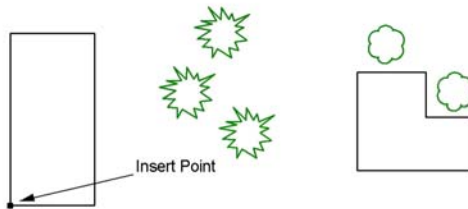


Figure 12-25. Data Selection for Translate/Rotate

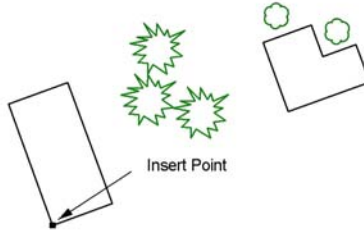



Figure 12-26. Data Rotated by 20 Degrees and Scaled to 0.75 of Original Size

Different procedures are available for:

- “Translate/Rotate Data” on page 12-23
- “Select Data” on page 12-25

Translate/Rotate Data

1. If no data was selected, click **Survey ► Translate/Rotate** or click the **Translate Rotate**  icon on the **Survey Tools** toolbar. The *Translate/Rotate* dialog box appears (Figure 12-27).

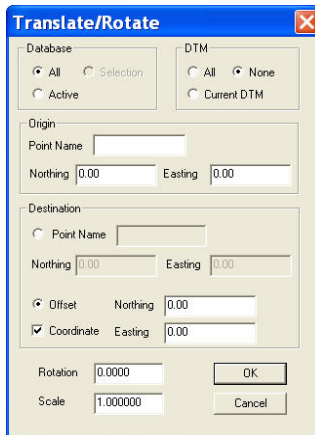


Figure 12-27. Translate/Rotate Dialog Box

2. Enter data into the *Translate/Rotate* dialog box, and then click **OK** to accept the changes. The data is moved, rotated, or scaled,

depending on the settings. For a description of the fields, see Table 12-6.


Table 12-6. Translate/Rotate Dialog Box Description

Field Name	Field Description
Database	<p>All: All of the data in the project (on active and inactive layers) are translated and/or rotated. This option is the default if there was no selection made before the command was started.</p> <p>Active: Only the data on the active layers will be translated and/or rotated.</p> <p>Selection: This is only available if a selection was made before the command was started.</p>
DTM	<p>This option is only active if no selections were made before the command was started.</p> <p>All: All of the DTMs in the project are translated and/or rotated.</p> <p>None: No DTMS are translated and/or rotated.</p> <p>Current DTM: Only the current DTM is translated and/or rotated.</p>
Origin	<p>Point Name: Input the number of the reference point in the dataset. All of the points will shift, rotate, or scale relative to this point.</p> <p>Northing/Easting: Input coordinates for the reference point in the dataset. This will only be allowed if a point name was not entered.</p>
Destination	<p>Point Name: Input the point number to move the dataset reference point or to use the Offset option. If you are only rotating or scaling data, then the origin point number can be entered again so the rotation or scaling occurs around the origin point.</p> <p>Northing/Easting: Input point coordinates to move the dataset reference point.</p>
Offset	<p>Select to nominate an offset for the translation instead of nominating a destination point.</p>
Coordinate	<p>Select this check box to define an Easting and Northing shift from the origin point.</p> <p>Input an X and Y shift in the adjacent Northing and Easting fields.</p> <p>If this check box is not selected, then the offset fields define the shift using a bearing and a distance.</p>
Rotation	<p>Enter the required rotation of data around the origin point or destination point, if a translation is not being made. The default is 1.0000 - no scaling.</p>
Scale	<p>Enter the scaling of the data around the origin point or destination point, if a translation is also being made. The default is 1.0000 - no scaling.</p>

Table 12-6. Translate/Rotate Dialog Box Description (Continued)

Field Name	Field Description
OK	Click to save.
Cancel	Click to cancel.

Select Data

1. Select the desired data.
2. Click **Survey** ► **Translate/Rotate** or click the **Translate Rotate**  icon on the **Survey Tools** toolbar.

The *Translate/Rotate* dialog box appears (Figure 12-28).

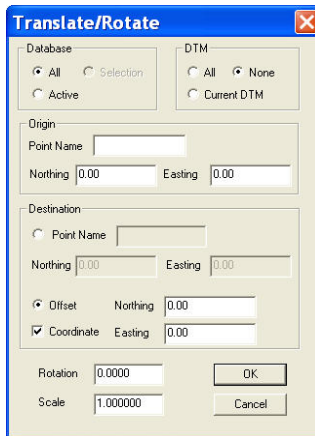


Figure 12-28. Translate/Rotate Dialog Box

3. Enter data into the **Translate/Rotate** dialog box, and then click **OK**. The data is moved, rotated, or scaled, depending on the settings. For a description of the fields, see Table 12-6 on page 12-24.
4. Click on a point within a selection to act as the origin point or enter the required information in the input bar.

Additional Information To move or translate the data from the origin point position to the new selected position with no change to rotation or scale values, leave the rotation set to 0.0000 and the scale set to 1.000.

- To rotate the selected data around the origin point, leave the offset as 0.000,0.000 and enter the rotation value. Leave the scale set to 1.000.
- To scale the selected data around the origin point, leave the offset as 0.000,0.000 and enter the scale value. Leave the rotation set to 0.0000.



Once a DTM has been translated, rotated, or scaled, it cannot be undone.

Traverse Adjustment

Use the Traverse Adjustment command to apply a traverse adjustment to a series of selected existing points and/or lines (Figure 12-29). The selected points or lines are adjusted and closed using the **Compass/Bowditch Method**.

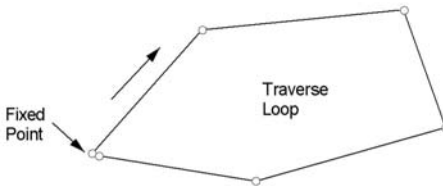


Figure 12-29. Initial Data Before Traverse Adjustment Completed

To use the Traverse Adjustment command, do the following:

1. Click **Survey** ► **Traverse Adjustment**.
2. Select the first point to define the traverse, or select the first line of the traverse closest to the second point on the traverse.
3. Select additional points or lines to complete the traverse selection.
4. Right-click and click **Adjust Traverse** from the shortcut menu. The **Traverse Adjustment** dialog box appears (Figure 12-30 on page 12-27).

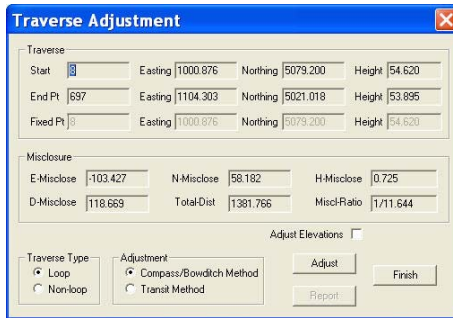


Figure 12-30. Traverse Adjustment Dialog Box

5. Modify the properties as required, and then click either **Adjust**, **Report**, or **Finish**. Table 12-7 describes the traverse adjustment fields.

Table 12-7. Traverse Adjustment Fields Description

Field Name	Field Description
Start	The coordinates and height of the traverse start point.
End Pt	The coordinates and height of the traverse end point.
Fixed Coord of End	If the traverse is a non-loop traverse, then enter the fixed coordinates of the traverse end point.
N-Misclosure	The misclosure in the Northing is displayed.
E-Misclosure	The misclosure in the Easting is displayed.
H-Misclosure	The misclosure in the height is displayed (if all of the points in the traverse have elevations).
D-Misclosure	The misclosure in the distance is displayed.
Total-Dist	The perimeter of the traverse is displayed.
Misc-Ratio	The accuracy of the misclosure is displayed as a ratio.
Traverse Type	Select the traverse type: Loop – Starts and ends on the same point. Non-Loop – Requires a fixed coordinate for the traverse end point.
Adjustment	Select either Compass/Bowditch Method or Transit Method for the adjustment.
Adjust	This button applies the adjustment to the traverse. Updates the points in the Survey View to their adjusted positions.

Table 12-7. Traverse Adjustment Fields Description (Continued)

Field Name	Field Description
Report	Creates a report of the traverse adjustment.
Finish	Closes the dialog box and returns to the Survey View.

LS (Least Squares) Network Adjustment

To perform a LS (Least Squares) Network Adjustment, click **Survey** ► **LS Network Adjustment**. The *Network Data Manager* dialog box appears (Figure 12-31).

On the *Network Data Manager* dialog box, perform the following to create a traverse or to open a new one:

- **New** – creates a new traverse (see Figure 12-32 on page 12-29).
- **Open** – opens an existing traverse.
- **Rename** – renames an existing traverse.
- **Delete** – deletes an existing traverse.
- **OK** – saves the traverse.

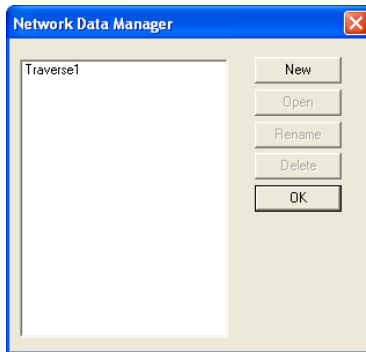



Figure 12-31. Network Data Manager

Click **New** on the *Network Data Manager* dialog box to display the **Data Editor** (Figure 12-32 on page 12-29).

The **Data Editor** has two panels: *Data Entry* and *Data Preview*. The Data Entry panel has two tabs: *Observations* and *Points*. The *Observation* tab contains the parameters for the measurement of raw data from the traverse. The *Point* tab contains the coordinate data for the points included in the traverse. Both tabs are configurable for content and order of data entry and edits. For detailed information on the Observations/Points tabs on the Data Editor, see “Using the Data Editor Observations and Points Tabs” on page 12-33.

To define the content and order of entry, use the context menu (click the right-mouse button on either the *Observations* or *Points* panel to display).

From the context menu, insert or hide columns for data entry. You can also specify the edit sequence of any element (type) by using the *Configure* dialog box. To edit, simply select any Type that displays in the Edit Sequence display and then manually enter the sequence, separating data by pressing the space bar. To set the order of data entry, drag and drop the headers until the desired order is achieved.

As data entry progresses, the preview pane updates to show the selected display options. Select the red square  at the bottom of the screen for the *Display Settings* dialog box to appear and specify options for points and numbers.

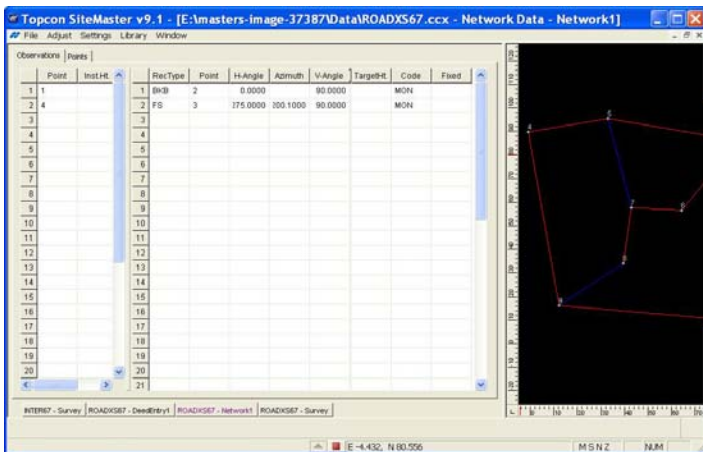


Figure 12-32. Data Editor

Managing/Processing LS Network Adjustments

On the *Data Editor* dialog box (see Figure 12-32 on page 12-29), use options from the **File** menu, the **Adjust** menu, the **Settings** menu, and the **Library** menu to manage and process the least squares adjustment.

File Menu Options Use options in the File menu to save and open traverse adjustments, generate reports, and to Import/Export files.

Table 12-8 describes the File menu commands for the Data Editor.

Table 12-8. Data Editor Parameters

File Menu	File Description
Save	Saves the traverse within the project. If the project has not been saved, the project is saved along with the traverse.
Open	Opens a traverse entry.
Save to Survey	Saves the traverse to the Survey view. Points and lines can be drawn, based upon survey codes when selected.
Regen	Regenerates the traverse, based upon the raw data. The regenerate report overwrites the traverse report. An option to overwrite the report will display when selected.
Report	Displays a report of the network adjustment.
Import	Imports existing third party least squares adjustment files, such as StarNet or xml files.
Export	Exports raw data files in an XML format.
Import Raw Data File	Imports Topcon tsj and tsv files as well as third party formats.
Survey View	Switches to the Survey view without processing the data.
Close	Closes the Data Editor.

Adjust Menu Option Use the **Network** option from the Adjust menu to perform a least square adjustment.

Settings Menu Options Use this option to specify the project weighting for the instrument, setup, and centering errors.

Click **Settings ▶ Instrument Settings**. The *Instrument Settings* dialog box appears (Figure 12-33 on page 12-31).

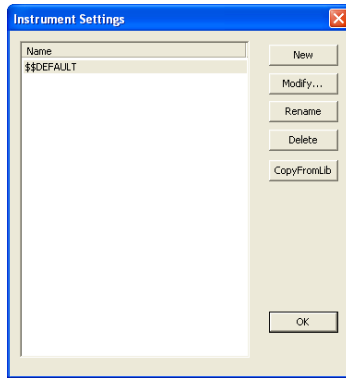


Figure 12-33. Instrument Setting Parameters

For a description of the Settings menu options, see Table 12-9 below.

Table 12-9.

Settings Menu	Menu Command Description
New	Creates new instrument settings for the project
Modify...	Modifies the settings of existing instrument settings
Rename	Renames existing instrument settings
Delete	Deletes instrument settings
CopyFromLib	Copies existing instrument settings from the instrument settings library

Click **New** or **Modify...** to display the *Library Settings Instrument* dialog box (Figure 12-34 on page 12-32) and do the following:

- Enter a **Name** and **Description** of the instrument and parameters for *Distance Constant*, *Distance PPM*, *Horizontal Angle*, *Direction*, *Azimuth/Bearing*, *Vertical Angle*, *Elev Diff Constant*, and *Elev Diff PPM*.



Distance PPM is typically derived from the EDM ratings being used. For example, an EDM with a rating of ± 6 millimeters and ± 3 ppm would be entered as 0.006 in metric units or as .019 in US Feet.



The Elev Diff PPM value amplifies the standard error on the elevations derived from the measurement of a zenith angle and a slope distance. Entering a value in this field causes larger total standard errors to be applied when longer slope distance measurements are involved.

- For **Centering Errors**, enter a Horiz Instrument and Horiz Target (ability to center over a mark). The entered values increases the standard errors of the horizontal angles, directions, and distance. Vertical centering errors relates to the estimate of vertical centering errors due to the inability to measure the exact instrument or target height from the ground.

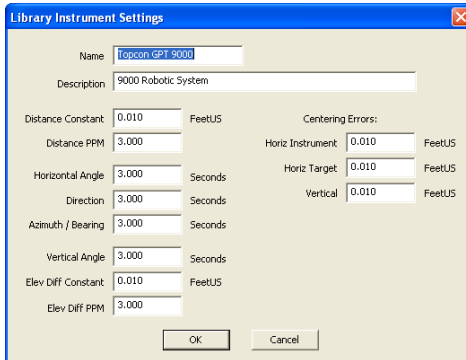


Figure 12-34. Library Instrument Settings Dialog Box

- Auto Recalc – if Auto Recalc is ON, new data or edited existing data will be automatically updated in the preview display; if Auto Recalc is OFF, you must select Regen from the File menu.



If repetitive edits are made to an individual measurement, a Regen may be required to establish the correct configuration of the traverse preview.

Library Menu The Library menu has the following options:

- Survey Codes – accesses the survey codes library
- Code Settings – accesses the code settings
- Instrument Settings – accesses the instrument settings library.

This option can also be used to create a new library entry and to modify, rename, delete, and/or copy an existing library entry. See Figure 12-33 on page 12-31.

Using the Data Editor Observations and Points Tabs

Information about the measurement raw data from the traverse can be found on the *Observations* tab and the *Points* tab contains the coordinate data for the points included in the traverse.

Observations Tab

The occupied point or setup point and the measurement data taken from the occupied point are the two panels that display on the *Observations* tab (Figure 12-35).

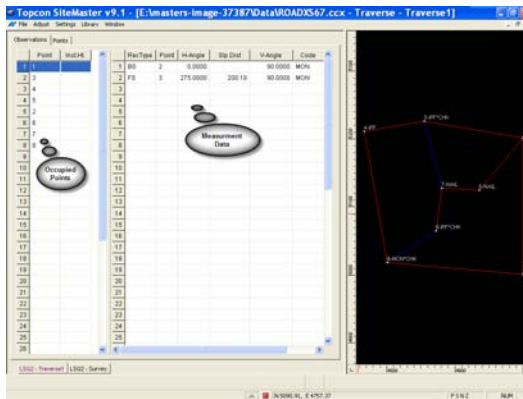


Figure 12-35. Observations Tab Information

Table 12-10 describes the *Occupied Point* panel fields.

Table 12-10. Occupied Point Pane Field Descriptions

Field Name	Field Description
Point	Defines the occupied point-by-point ID.
Inst. Ht.	Sets the instrument height for the occupied point.

Table 12-10. Occupied Point Pane Field Descriptions (Continued)

Field Name	Field Description
Inst. Settings	Sets the instrument settings to be used for the setup. Different settings can be used for various set ups in a project. Where a specific instrument setting is not specified for a setup, the default instrument settings will be used.
Station ID	Setup numbering begins at the first setup and numbers sequentially thereafter. This number may or may not match the Occupied (Setup) point number.
Swing	Enter the Swing angle to be applied to all observations from the station point. Default is 0.000.

Figure 12-36 below shows an example of an *Occupied Point Field* specifying instrument settings.

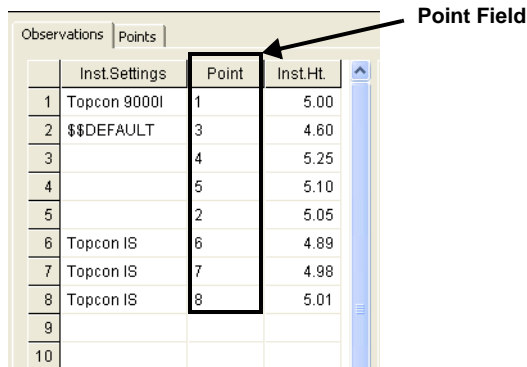


Figure 12-36. Example – Occupied Point Field with Specified Instrument Settings

The *Point* field is configurable for content and order of data entry and edit (Figure 12-36). To define the content for entry, press and hold the mouse, then right-click to display the **Context** popup menu (Figure 12-37 on page 12-35).

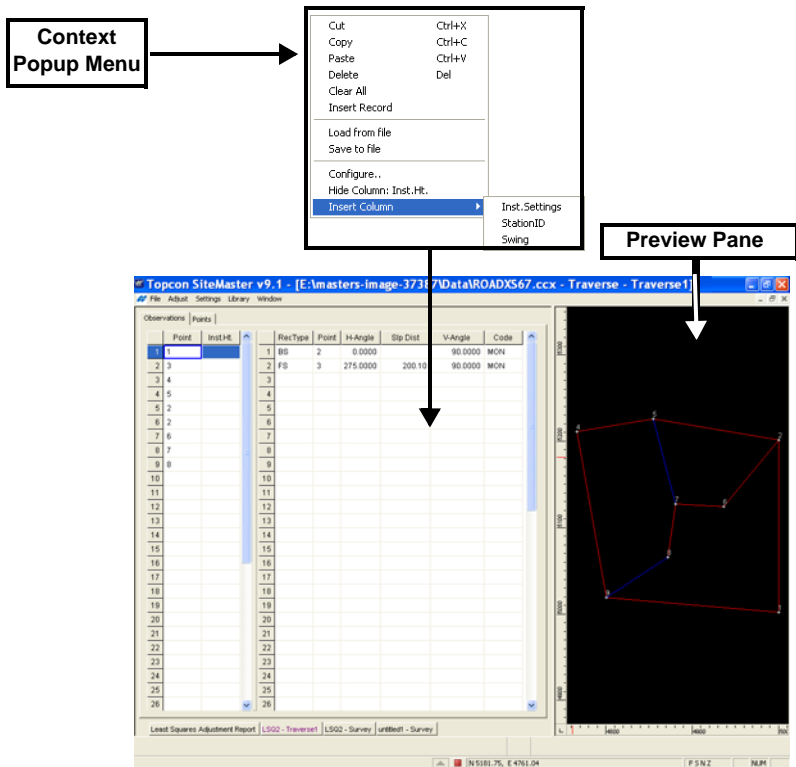


Figure 12-37. Context Pop-up Menu on the Data Editor Dialog Box

From the **Context** pop-up menu you can do the following:

- Insert or Hide columns for data entry.
Enter parameters in the **Display Sequence** field from the **Configuration** dialog box (Figure 12-37). However, it is easy to drag and drop the headers to their desired location using the Hide Columns/Insert Columns commands on the Context pop-up menu.
- Specify the editing sequence of any element (type) by using the **Configure** menu option (Figure 12-38 on page 12-36) on the Context pop-up menu (Figure 12-37) and then selecting any option under the **Edit Sequence** panel.

To edit, simply select the type and then manually enter the sequence, using the space bar to separate data.



The list of available elements to Add or Hide depends upon the current display.

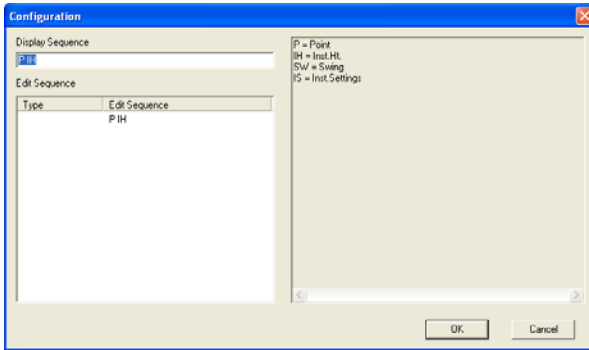



Figure 12-38. Configuration Parameters

As data is entered, the preview pane is updated to show the selected display options. To specify the display options, click on the red square  at the bottom of the screen to display the **General Settings** dialog box.

Selecting a Record Type Correctly selecting the record type (RecType) is crucial to the processing of the adjustment. Each tab is configurable for content and order of data entry and edits.

Table 12-11 describes the record types.

Table 12-11. Adjustment Record Types

Record Type	Record Type Description
BKB	Defines a backsight by direction.
BS	For backsighting between two points.
FS (Foresight Record)	Contains measurement data to a foresight. A new setup definition is expected after this record.
PT (Point Record)	Contains the coordinates of a point.

Table 12-11. Adjustment Record Types (Continued)

Record Type	Record Type Description
SS (Sideshot Record)	This record type does not expect a new setup definition when used. Numerous sideshots can be defined from an individual setup.

The header types and their definitions are described in Table 12-12 below.

Table 12-12. Adjustment Header Types

Header Types	Header Description
Point	The occupied point ID
H-Angle	Enter the horizontal angle from the set up station to the observed point on this line.
Slp Dist	
V-Angle	Enter the vertical angle from the set up station to the observed point on this line. Displays or allows entry of the slope distance measured between two points.
Code	Enter a code for the observed point. This code may be a feature code from the Survey Code library. String numbers and comments may also be used. The code is carried forward from one line to the next during data reduction, so codes only need to be entered when they change.
Azimuth	Enter the Azimuth of the measurement. (Optional)
Bearing	Enter the Bearing of the measurement. (Optional)
Fixed	Toggle this option On to hold the measurement data as fixed between two points. This data will not be modified during adjustment. To toggle this option to ON, be sure you are in the measurement row to be held, then double-click on the cell under the fixed header. You can also toggle ON/OFF by highlighting the cell, then pressing the spacebar.
Horz Dist	Horizontal distance from the occupied point to the target.
Ht. Diff	Elevation difference between the occupied point and the target.
TargetHt	The target height above the surface.
Vert Dist	The vertical distance between the Instrument and the target.

Both the *Observations* tab and the *Points* tab are configurable for content and order of data entry and edit. To define the content for

entry, access the Context popup menu by clicking and holding down the right mouse button (Figure 12-39).

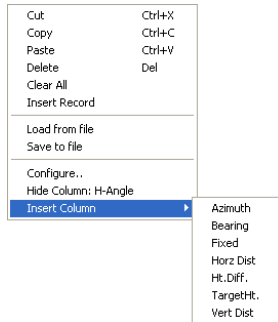


Figure 12-39. Insert a Column using the Insert Column Command

From the Context popup menu you can do the following:

- Insert or Hide columns for data entry.
- Enter parameters in the *Display Sequence* field from the *Configuration* dialog box (Figure 12-40). However, it is easy to drag and drop the headers to their desired location using the *Hide Columns/Insert Columns* commands on the Context popup menu.
- Specify the editing sequence of any element (type) by using the **Configure** menu option (Figure 12-39) on the **Context** popup menu and then selecting any option under the *Edit Sequence* panel.

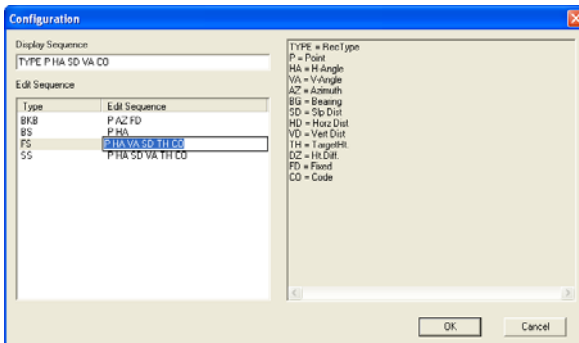


Figure 12-40. Using the Edit Sequence Panel


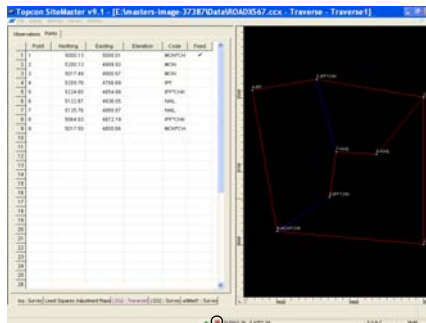
As data is entered, the preview pane is updated to show the selected display options. To specify the display options, click on the red square  at the bottom of the screen to display the *General Settings* dialog box.

Table 12-13 below lists the **Context** popup menu options and their description.

Table 12-13. Context Popup Menu Options

Menu Option	Menu Description
Cut	Cuts a record
Copy	Copies a record
Paste	Paste a record
Delete	Deletes selected record or cell contents
Clear All	Clears all cells
Insert Record	Inserts a record above the row of the active cell
Load from file	Loads points or observations from a file
Save to file	saves selected records to a text file
Configure..	Displays the Configuration dialog box
Hide Column	Hides a column on the active cell
Insert Column	Inserts a column on the active cell

Displaying Points Tab Information The Points tab displays information about points, coordinates, codes, and so on (Figure 12-41).



Hot Button Icon

Figure 12-41. Points Tab Options

The Point field is configurable for content, order of data entry, and edit. To define the content for entry, press and hold down the right mouse button to access the Context popup menu (Figure 12-42).

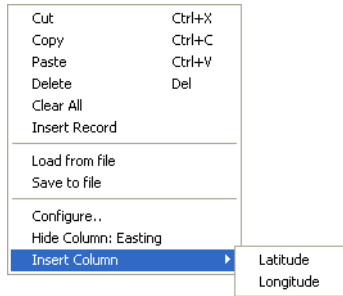


Figure 12-42. Context Popup Menu

Click on the *Configuration* menu option (Figure 12-43) to specify the edit sequence of any element (type). You can select any option under the *Edit Sequence* panel.

To edit, simply select the type and then manually enter the sequence; separate data by pressing the spacebar.

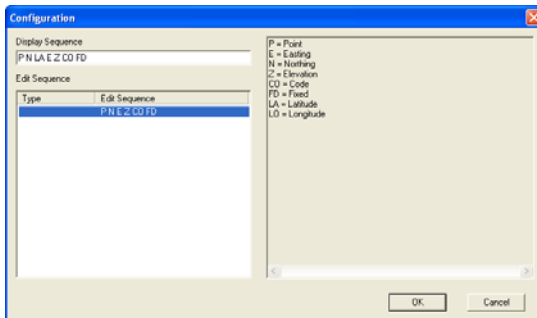


Figure 12-43. Points Tab Configuration Screen

The display sequence can also be defined or edited from the *Configuration* dialog box. However, it is very easy to drag and drop the headers to their desired location.

Point coordinates can be “Fixed” by placing a check mark in the Fixed column for a point. This prevents the coordinates of the point from being altered during adjustment. If the fixed column is not

displayed, right-click within the spreadsheet area and choose to insert the *Fixed* column.

Least Squares Transformation

Use the Least Squares Transformation command to transform selected data from one coordinate system or datum to another using a Helmert or Least Squares Transformation. Ensure that at least three corresponding points exist in each coordinate system within the project data. This data may be displayed in two Survey panes of the same project, as shown in Figure 12-44 and Figure 12-45.

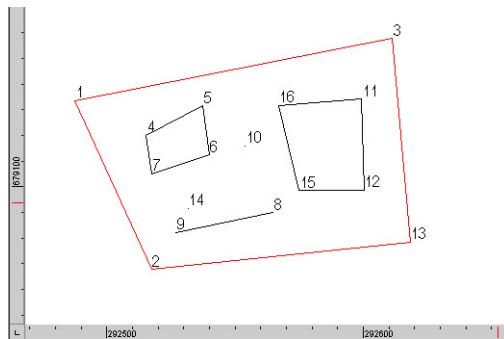


Figure 12-44. Data in Original Datum – Window 1

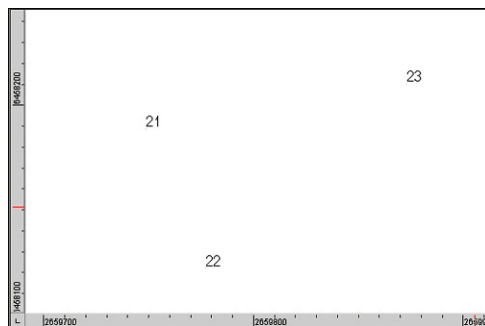


Figure 12-45. Three Known Fixed Points in the New Datum - Window 2

Do the following to use this command:

1. Click **Survey** ► **LS Transformation**.

The *LS Transformation* dialog box appears (Figure 12-46).

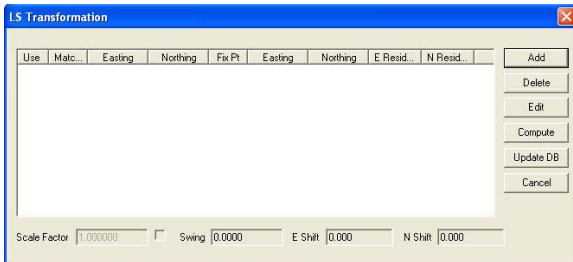


Figure 12-46. LS Transformation Window

2. Click **Add**.
3. Click to select a point in the original coordinate system that will be transformed, or enter its point number in the input bar (Figure 12-47), and press **Enter**.



Figure 12-47. LS Transformation Input Bar

4. Click to select the corresponding point in the new coordinate system and press **Esc**, or enter its known point number in the input bar, and press **Enter**.

If no data was selected, then all of the data in the dataset will be selected.

5. Repeat steps two to four until you have selected all of the required points. The coordinate information is updated in the Transformation panel (Figure 12-48). For a description of the Transformation fields, see Table 12-14 on page 12-43.

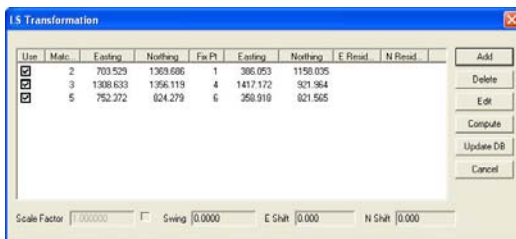


Figure 12-48. Transformation Panel

6. To transform data without scaling it, select the **Scale Factor** check box and enter a value of 1.

7. Click **Compute** to calculate the transformation parameters.
8. Analyze the residuals, and note any errors that may give an unacceptable error when the transformation is implemented.
9. To delete a line, select it and click **Delete**.
10. To edit a line, select it and click **Edit**.
The *LS Edit* window appears (Figure 12-49).

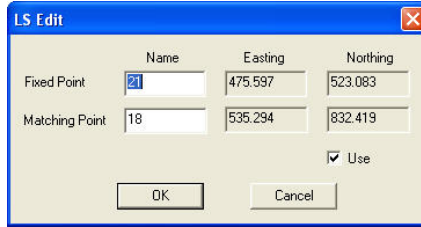


Figure 12-49. LS Edit

11. Make the required changes, and click **OK**.
12. Click **Apply** to complete the Least Squares transformation.
The selected data is moved to its position in the new coordinate system, and the data is displayed at this new position in the Survey View. The transformation result is displayed in a new dialog box.

Table 12-14 describes the fields.



If any data is edited or removed from the computation, it is essential that the transformation is computed again using the **Compute** button.

Table 12-14. LS Transformation Window Field Descriptions

Field Name	Field Name Description
Use	Click to clear the check box of a line that may have an error. This removes the line from the calculation.
Match Point	Displays the original point number.
Easting	Displays the Easting coordinate of the original point number.
Northing	Displays the Northing coordinate of the original point number.
Fix Pt	Displays the corresponding fixed point number in the new coordinate system.

Table 12-14. LS Transformation Window Field Descriptions (Continued)

Field Name	Field Name Description
Easting	Displays the Easting coordinate of the fixed point number.
Northing	Displays the Northing coordinate of the fixed point number.
E Residual	Displays any residuals in the Easting.
N Residual	Displays any residuals in the Northing.
Scale Factor	Displays the calculated transformation scale factor.
Swing	Displays the calculated transformation swing.
N Shift	Displays the calculated shift in the Northing.
E Shift	Displays the calculated shift in the Easting.

Define Structure

Use the Define Structure command to outline a structure and save the selected points, lines, and arcs to a nominated file name. The data may represent a structure outline or other feature, such as a dam, cul de sac head, traffic island, or roundabout.

To use the Define Structure command, do the following:

1. Click **Survey ► Define Structure**.
2. Click to define one corner of the window, which will select all of the data in the window of the structure.

The *Save As* dialog box appears (Figure 12-50).

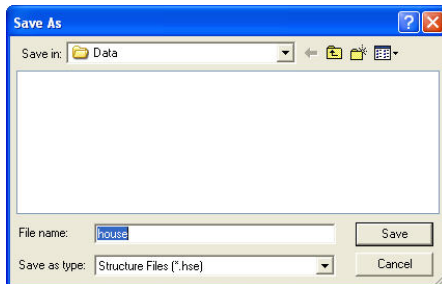


Figure 12-50. Define Structure

3. Type a name, use the *.hse* file extension, and click **Save**.

4. Click to define the diagonal corner of the window to complete the data selection.
5. Click on a point within the window to act as the insertion and rotation point when the structure is located in a project (Figure 12-51).

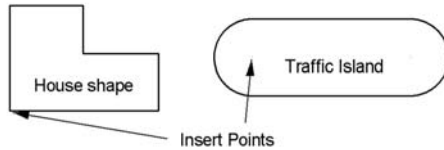


Figure 12-51. Defining Structure Illustration

Locate Structure

Use the Locate Structure command to create points, lines, and arcs representing a structure or other object previously saved with an *.hse* extension. The object may be moved and manipulated so it is positioned exactly at the required rotation and offset correctly from boundaries using a range of functions.

To use the Locate Structure command, do the following:

1. Click **Survey ► Define Structure**. The *Open* dialog box appears (Figure 12-52).

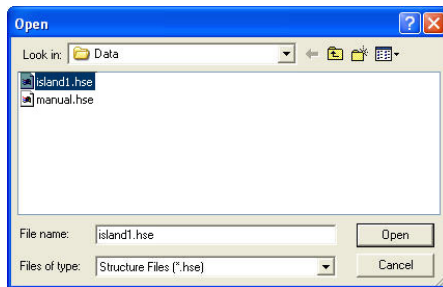


Figure 12-52. Locate Structure

2. Select the desired file, and click **Open**.
The structure is displayed on the screen with the pointer attached to the insertion point. A toolbar also appears.

3. Position the insertion point, and left-click the mouse.
The structure is positioned at this point, but may be moved.
4. Right-click to select options from the shortcut menu or select a tool from the toolbar. For a toolbar description, see Table 12-15 on page 12-46 below.
5. To load another .hse file, click **Load** on the input bar (Figure 12-53).

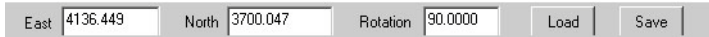











Figure 12-53. Locate Structure Input Bar

6. To save a structure in the current position, click **Save** on the input bar.

Table 12-15. Tools Descriptions

Tools Name	Tools Name Description
Direction 	Rotates the structure around the insertion point to a nominated bearing. The default bearing is 0.0000.
Swing 	Rotates the structure around the insertion point to a nominated angle.
Parallel 	Rotates the structure with a line of the structure parallel to a nominated line in the project with an optional offset setting between the lines.
Corner 	Snaps the insertion point to a selected point in the project. Any current rotation applied to the structure is held.
X Mirror 	Creates a mirror image of the structure around the X-axis of the structure.
Y Mirror 	Creates a mirror image of the structure around the Y-axis of the structure.
Offset to Line 	Offsets the insertion point from a line or if offsets are displayed to several boundary lines, it sets the offsets of the closest point from a boundary.
Display Offsets 	Displays the lines or arcs from which the offset of the closest point of the structure is calculated.
Save 	Saves the structure in the current position, creating the points, lines, and arcs on the current layer with properties as set on the property bar.

Survey View: Design Menu

This section describes the different Design View menu commands on the Design menu located on the main menu bar (Figure 13-1).

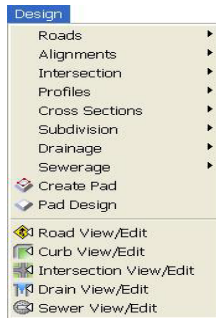


Figure 13-1. Survey View – Design View Menu

In This Section:

- “Manual Road Entry”
- “Level Book” on page 13-12

Manual Road Entry

Use the Manual Road Entry command to enter the data for a Road from data provided in a manual (non-electronic) format. You can create roads data consisting of a horizontal alignment, profile(s), and offset strings, or from a horizontal alignment and cross sectional data.

The horizontal alignment and surface data is saved to Survey View and the roads data with profile, strings, and cross sections saved as a Design surface in Roads View.

To enter data for a new road, do the following:

1. Click **Design ▶ Roads ▶ Manual Road Entry**. The *Manual Road Entry Manager* dialog box appears (Figure 13-2).

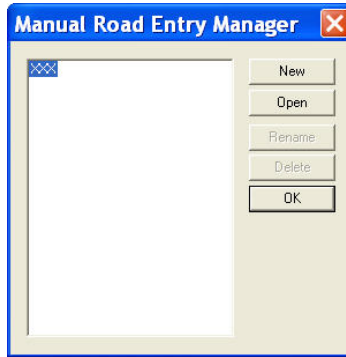


Figure 13-2. Manual Roads Entry

2. Do the following to create a manual road entry:
 - **New** – press to create and open the dataset for a new road. The first road will be allocated the default name *ALIGN1*. This can be renamed later if you prefer. The *manual road entry view* dialog box appears (Figure 13-3 on page 13-3).
 - **Open** – press to open a road entry file.
 - **Rename** – press to rename a highlighted road entry file.
 - **Delete** – press to delete a highlighted road entry file.
 - **OK** – press to save road entry parameters.

The Road Entry view consists of two pull-down menus, four pages for entry of different types of road data, and a context menu that displays by right-clicking the mouse (Figure 13-3 on page 13-3).

Roads data consists of an alignment and cross sections (using the Alignment and Cross Section tabs) or an alignment and a number of strings with profiles (VCs) using the Alignment, Profile, and Offset String tabs.

Alignment tab All roads that are manually defined by this function starts with the definition of a horizontal alignment. To check these options, click **File ▶ Configure** from the *manual road entry view* dialog box (Figure 13-3 on page 13-3).

You configure the alignment to suit the types of manual data available for the definition of the alignment.



The different columns of data may be on seen or hidden and the various columns may be moved to a different position by dragging the header of a column with the mouse.

- Enter the horizontal alignment data using a combination of its component entities as records in order from the start point of the alignment.
 - Start (Point): defined by a coordinate position (Easting, Northing) and the start station (running distance) of the alignment.
 - Straight: straight line defined by a bearing and a length.
 - Arc: defined by a radius and an arc length. A negative radius will create a curve to the left (clockwise).
 - Spiral: defined by its length (if required).

As data is entered, the station distance and the Easting/Northing coordinates will be computed. The window to the right graphically displays the alignment entered (Figure 13-3).

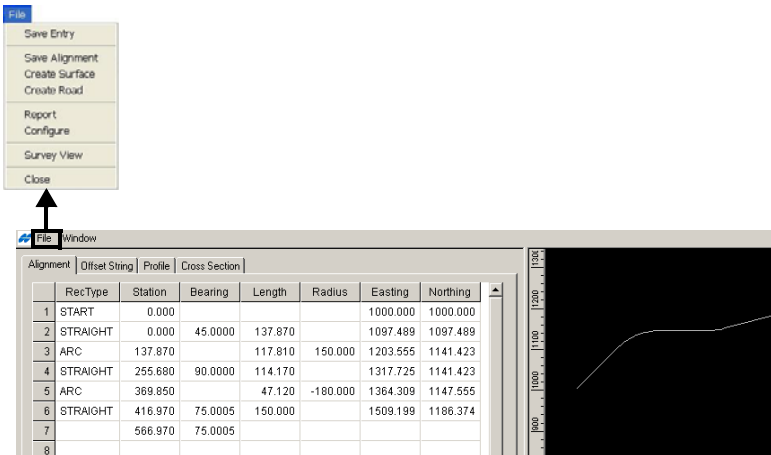


Figure 13-3. Manual Road Entry View Parameters

When entering a spiral length before and after an arc, use the ‘RecType’ SPIRAL and enter its length in the ‘*Length*’ field as shown in Figure 13-4.

RecType	Station	Bearing	Length	Radius	Easting	Northing
1	START	0.000			1000.000	1000.000
2	STRAIGHT	0.000	45.0000	137.870	1097.489	1097.489
3	ARC	137.870		117.810	150.000	1203.555
4	STRAIGHT	255.680	90.0000	102.657	1306.212	1141.423
5	SPIRAL	358.337		23.000	1329.203	1141.912
6	ARC	381.337		24.119	-180.000	1353.098
7	SPIRAL	405.456		23.000	1375.432	1150.535
8	STRAIGHT	428.456	75.0005	138.490	1509.204	1186.375
9		566.946	75.0005			
10						

Figure 13-4. Entering Spiral Length Parameters



The Alignment IPs and TPs will display in the graphics window if activated pressing the **Hot** (red) Button at the bottom of the screen.

- Define the alignment using its Intersection Points. In this case, the above data could be defined as:
 - Start Point: Defined by a coordinate position (*Easting/Northing*) and the start station (running distance) of the alignment.
 - IP: Defined by bearing and length from the previous IP station. Add the radius of the arc to this record (Figure 13-5).

RecType	Station	Bearing	Length	Radius	Easting	Northing
1	START	0.000			1000.000	1000.000
2	IP	0.000	45.0000	200.000	1141.421	1141.421
3	IP		90.0000	200.000	-180.000	1341.421
4	IP		75.0005	173.700	1509.204	1186.375
5		566.946	75.0005			
6						

Figure 13-5. Alignment Defined Using Intersection Points

When using Spiral Curves, activate the two *Spiral Length* columns from the context menu. Then enter the spiral lengths to the IP definition (Figure 13-6 on page 13-5).

File Window									
Alignment Offset String Profile Cross Section									
	RecType	Station	Bearing	Length	Radius	Spiral L1	Spiral L2	Easting	Northing
1	START	0.000						1000.000	1000.000
2	IP	0.000	45.0000	200.000	150.000			1141.421	1141.421
3	IP		90.0000	200.000	-180.000	23.000	23.000	1341.421	1141.421
4	IP		75.0005	173.700				1509.204	1186.374
5		566.946	75.0005						
6									
7									

Figure 13-6. Alignment Tab Parameters

It is not necessary to enter crossfalls. The **Cross Fall** column may be left blank; elevations can be computed from a profile.

After all alignment data has been entered, do the following:

1. On the *manual road entry view* (Figure 13-3 on page 13-3), click **File ▶ Save Entry** to save the entered data (Figure 13-7).
2. On the *manual road entry view* (Figure 13-3 on page 13-3), click **File ▶ Save Alignment** to save alignment data to Survey View. The alignment is created with appropriate data applied to each horizontal intersection point (Figure 13-7).



Figure 13-7. Save Data Entry and the Alignment

Offset String If the manual data includes the string details that are offset from this alignment and/or from each other, enter them on the *OffSet String* tab.



Be sure that an alignment has been created before continuing with this option.

- Each string will be offset from the original alignment or from a previously entered string. The usual data entered for a string consists of its String Name, its Reference String (default is CL).

and the offset and crossfall from the Reference String at each Station position. There is an alternative option to define the string using a difference in elevation. Activate the “Elev Diff” column, if necessary.

- Set each string to the Left or to the Right of the Reference String. If the string is at a constant offset and crossfall from the reference string, only one line of data is required. The new string displays on screen (Figure 13-8).

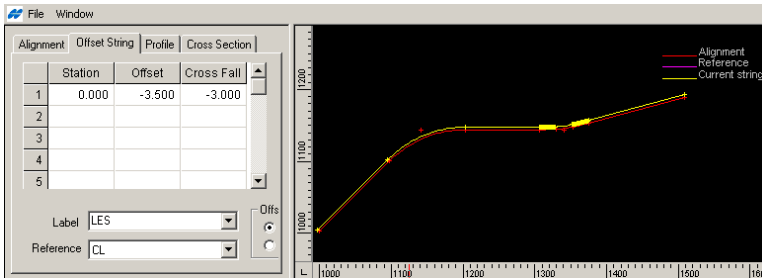


Figure 13-8. Offset String Tab Parameters

- If the offset distance and the crossfall varies along the length of the string this can be entered into their respective data fields. In this case, the rate of change of offset and the crossfall is taken between any two consecutive entries. To hold the offset and crossfall from one station to another, repeat the entry as shown in Figure 13-9 on page 13-7.

It is not necessary to enter both offset and cross fall values on every record if only one value changes.

	Station	Offset	Cross Fall
1	0.000	3.500	-3.000
2	320.000	3.500	-3.000
3	330.000	3.750	-4.000
4	420.000	3.750	-4.000
5	430.000	3.500	-3.000
6			
7			

Label:
 Reference:

Offset: Left Right

Figure 13-9. Enter Offset and Crossfall Parameters

It is unnecessary to enter crossfalls. The cross fall column may be left blank; elevations can be computed from a profile.

Profile Tab On the Profile tab, at least one string must have a defined profile. Enter this data on the Profile tab (Figure 13-10 on page 13-8).



An alignment must be created before defining a profile.

The usual data entered for a profile consists of Station and Level and Curve Length data. Other data entry columns are available for Grade and K Factor. A profile is entered for the selected string. This is usually the CL string, but further profiles may be entered for other strings already defined using “Offset String”. Select the current string from the ‘String’ drop-down box.

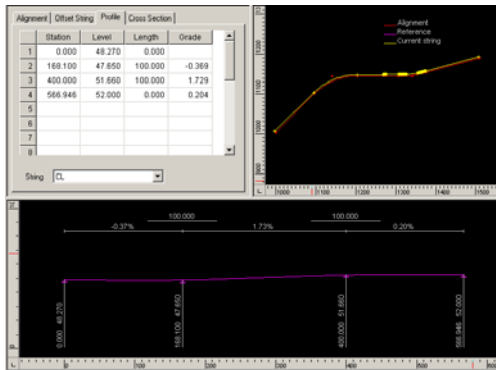


Figure 13-10. Profile Tab

As the profile data is entered, it will graphically display in the profile window to the right. Use the ‘Hot Button’ to add other details to the profile view and to set an exaggeration.

Cross Section Tab On the Cross Section tab, enter cross section levels and offset/cross falls data for the road (Figure 13-11 on page 13-9). If using the ‘Cross Section’ data, then data is not entered into the ‘Profile’ and ‘Offsets Strings’ tabs. The profile is built from the 0.00 offset levels entered on this tab.



An alignment must be created before defining the cross fall.

As cross section data is entered, it graphically displays in the cross section window to the right. Use the ‘Hot Button’ to add other details to the cross section view.

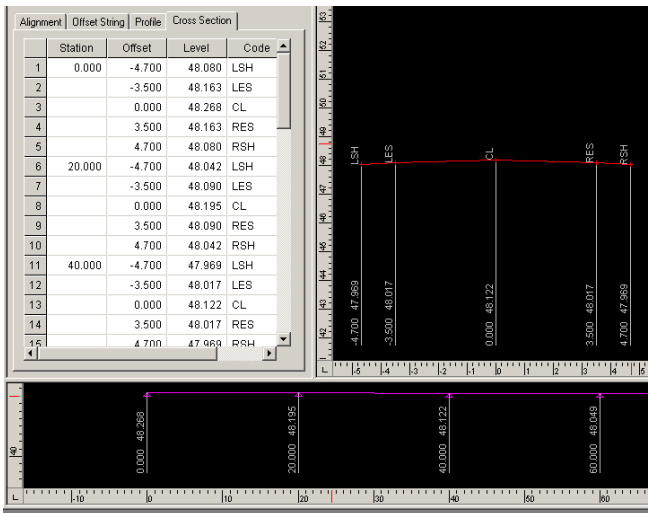


Figure 13-11. The Cross Section Tab

Create Surface

An alignment, a profile, and offset strings must be created before creating a surface. Alternately, the entry will contain alignment and crosssection data.

To create a surface, click **File** ► **Create Surface** from the *manual road entry view* dialog box in Survey View (Figure 13-3 on page 13-3).

The *Create Surface* dialog box appears (Figure 13-12 on page 13-10).

Do the following to create a DTM surface:

- Default spacing for the mesh created along straights and through curves is shown.
- Select either the *Cross Section template* or the *Offset string* option from which to calculate data.
- Check mark the *Save DTM* box and enter a *DTM name* for points defining the DTM and the strings.

- On the **Transfer Table** tab, define the Code for points, string names, and the layers created in Survey View.
- You may enter a **Road number** and press **Append** to add it to the string names.
- Click **OK** to create the DTM surface in Survey View.

A surface model is created and layers defined to hold the points and string data.

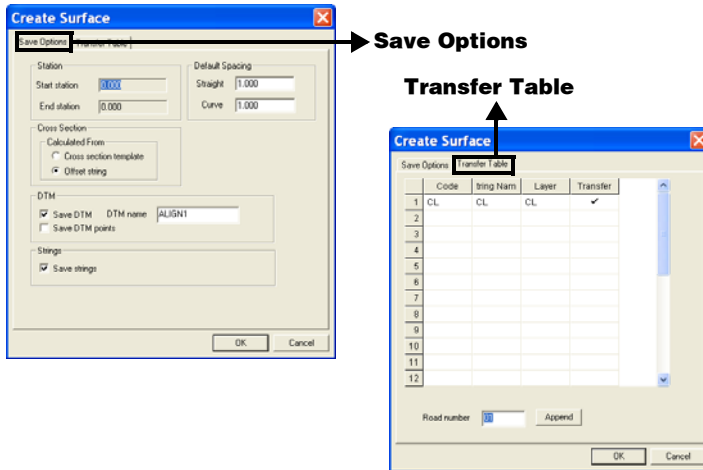


Figure 13-12. Create a DTM Surface

Create Road

Before creating a cross section, an alignment, a profile, and offset strings must be created.

To create a road, click **File** ► **Create Road**. The **Create Cross Section** dialog box appears (Figure 13-13 on page 13-11).

Do the following to create a cross section:

- Enter a Layer name from the drop-down entry box.
- The entered data forms a Design View in Roads View. A natural surface can be extracted from a suitable DTM in the current project.

- Select either the *Cross Section template* or the *Offset string* option from which to calculate data.
- Check mark the *Include TPs and IPs* box to include in the data.
- Default spacing for the cross sections created along straights and through curves is shown; new parameters can be entered.
- On the Additional Stations tab, specify the enter the required between the specified start and end station.
- Enter a station value in the *Start Station* column to add in one extra station.
- To omit a station, enter the station value in the *Start Station* column and place a check mark in the *Omit* column.

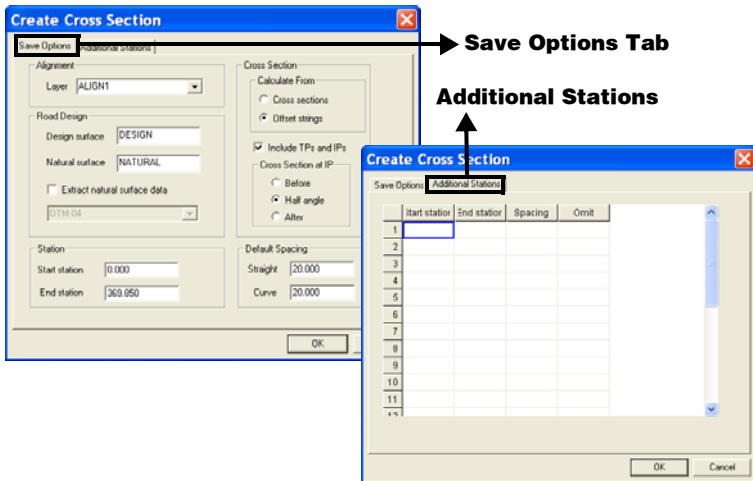


Figure 13-13. Create a Cross Section

Report To create a report of the entered data, click **File ► Report**. This report can be saved to a file or printed out for future reference.

Configure To modify the configuration of the columns available on the 4 individual tab, make a tab active and click **File ► Configure** or click **Configure** from the context menu (Figure 13-14).

The *Display Sequence* field shows the order of the columns and of those that display on the screen to the right. The column order may be changed by dragging the column name to a new position. Additional

columns may be inserted using the Insert Record option on the context menu.

The *Edit Sequence* field displays the order that data entry will progress using the Enter key in a record. It can be altered on this screen or by using the Edit Sequence option on the context menu to click the TAB on each column in the selected order.

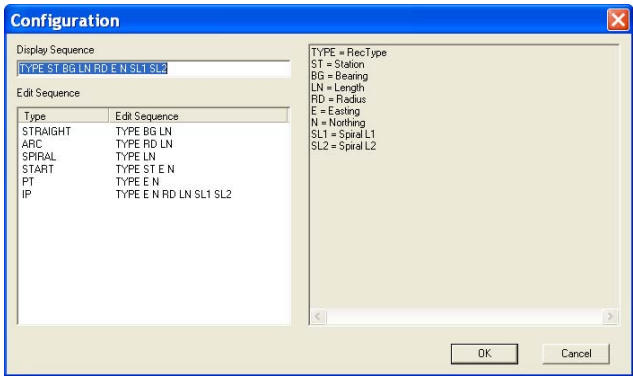


Figure 13-14. Modify the Configuration of Data

Level Book

The Level Book command allows you to enter cross sectional survey data in the form of staff reading, chainage, and offset to be used in road design. Reduced levels can also be entered here.

To enter cross sectional survey data, do the following:

Click **Design** ▶ **Roads** ▶ **Level Book**. The *Level Book Manager* dialog box appears (Figure 13-15 on page 13-13).

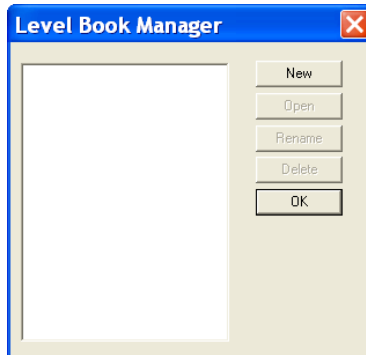


Figure 13-15. Design Menu – Level Book

Press New. The *level book entry* table appears (Figure 13-16).

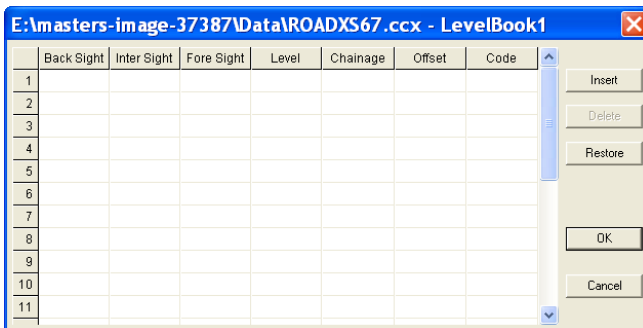


Figure 13-16. The Level Book Entry Dialog Box

Table 13-1 describes the level book entry table fields.

Table 13-1. Level Book Entry Field Descriptions

Field Name	Field Descriptions
Back Sight	Enter the observed reading to the backsight.
Inter Sight	Enter observed reading to the intermediate sight
Fore Sight	Enter the observed reading to the fore sight.
Level	Enter known reduced levels only at the appropriate back sights.
Chainage	Enter the chainage or distance for the observation. This field only needs to be entered when the chainage changes.

Table 13-1. Level Book Entry Field Descriptions (Continued)

Field Name	Field Descriptions
Offset	Enter the offset of the observation point from the centerline. Negative values are taken as being to the left of the centerline in the direction of increasing chainage.
Code	Enter an alpha-numerical code for the observation. This field is optional.
Insert	Inserts a row.
Delete	Deletes a highlighted row.
Restore	Restores a deleted row.
OK	Saves cross sectional data entered.
Cancel	Quits the routine without saving.



The Code column is used to assign a description to each observation and is optional. The codes can be used in longitudinal section plotting and in the conversion of a road into a digital terrain model. The code need only be entered when it changes.

Defining Alignments

An alignment is usually the center line of a road, pipeline, service, or other feature. It is often used to produce profile and cross section data. An alignment consists of a number of intersection points (IP) joined together with lines. An IP is linked to a point in the database. If it represents a curve in a road, then the IP may have arc and spiral parameters defined. It is not necessary to create any line segments before creating an alignment, although this may be helpful. Existing line or string data may be converted to an alignment.

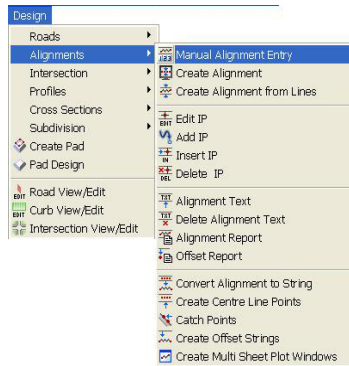


Figure 13-17. Design Menu – Alignments Options

In This Section:

- “Create Alignment” on page 13-20
- “Edit IP (Intersection Points)” on page 13-25
- “Manual Alignment Entry” on page 13-16
- “Edit IP (Intersection Points)” on page 13-25
- “Add IP” on page 13-28
- “Insert IP” on page 13-29
- “Delete IP” on page 13-30
- “Alignment Text” on page 13-30
- “Delete Alignment Text” on page 13-32
- “Alignment Report” on page 13-32
- “Offset Report” on page 13-34
- “Convert Alignment to String” on page 13-35
- “Create Center Line Points” on page 13-36
- “Catch Points” on page 13-38
- “Create Offset Strings” on page 13-40
- “Create Multi Sheet Plot Windows” on page 13-41

Manual Alignment Entry

Use the Manual Alignment Entry command to define a horizontal alignment from provided data that defines the full geometry of the alignment in terms of the lines, arcs, and spirals that make up the horizontal alignment. It is different from the **Create Alignment** command that defines the frame of the intersection point (IP) positions for an alignment. The manual alignment entry includes the entities directly into the alignment so the IPs are simply points at which there is a tangential connection between one entity and the next.

To use the Manual Alignment Entry command, do the following:

1. Set the current layer and appropriate parameters for the new point and alignment data.
2. Click **Design** ▶ **Alignments** ▶ **Manual Alignment Entry** (Figure 13-17 on page 13-15).

The *Manual Alignment Entry Manager* dialog box appears (Figure 13-18).

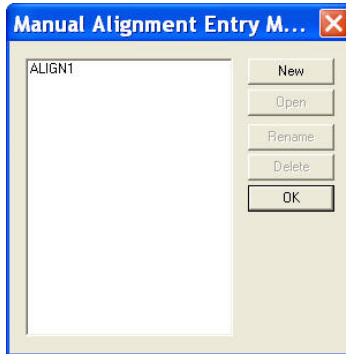


Figure 13-18. Manual Alignment Manager

3. Click the **New** button on the *Manual Alignment Entry Manager* dialog box (Figure 13-18 on page 13-16). The data entry/editor dialog box appears. Table 13-2 describes the Manual Alignment Entry Manager buttons .

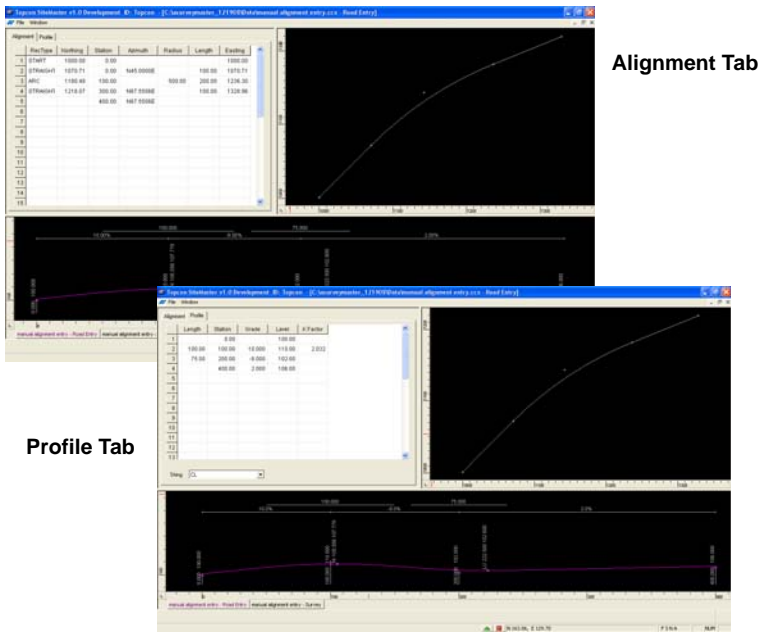
Table 13-2. Manual Alignment Manager Field Descriptions

Field Name	Field Descriptions
New	Creates a new alignment in the project.
Open	Opens a previously saved manual alignment. Highlight the name, and click Open .
Rename	Renames an alignment. Highlight its name, and click Rename . Type its new name.
Delete	Deletes an alignment. Highlight its name, and click Delete . This deletes the alignment and all of the data currently entered in this option.
OK	Cancel the routine and closes the Manager.

When a new alignment is created, a *data entry/editor* dialog box appears (Figure 13-19 on page 13-18) and an alignment name, starting with *ALIGN1* and then *ALIGN2* is allocated.

The *data entry/editor* dialog box is separated into three sections and two entry tabs:

- The spreadsheet – for data entry (top Left)
- The Alignment preview pane – section to edit horizontal alignments.
 - Alignment tab: edits horizontal alignments (centerlines).
- The Profile preview pane – section to edit vertical alignments.
 - The *Profile* tab: edits vertical alignments (profiles).



Profile Tab

Alignment Tab

Figure 13-19. Manual Alignment Entry – Alignment/Profile Tabs

Each preview pane has zooming/editing options accessed from the **Context** popup menu (right-click on the mouse). For Display Setting options, click on the **Hot** button icon ■ at the bottom of the screen (Figure 13-19).

Both the *Alignment* tab and the *Profile* tab are configurable for content and order of data entry and edit.

To define the content and order of entry, right-click the mouse on the spreadsheet (Figure 13-19) to display the **Context** popup menu (Figure 13-20 on page 13-19).

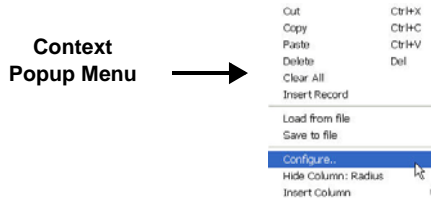


Figure 13-20. The Context Popup Menu – Configure Menu Option

The File menu on the *data entry/editor* dialog box (Figure 13-19 on page 13-18) has the following options:

- **Save Entry** – saves both the data entry and the project.
 - **Save Alignment** – saves the alignment to the survey view.
 - **Report** – generates a horizontal alignment report.
 - **Survey View** – switches to the Survey View.
 - **Close** – closes the Alignment editor.
4. Click **Configure**. The *Configuration* dialog box appears (Figure 13-21).

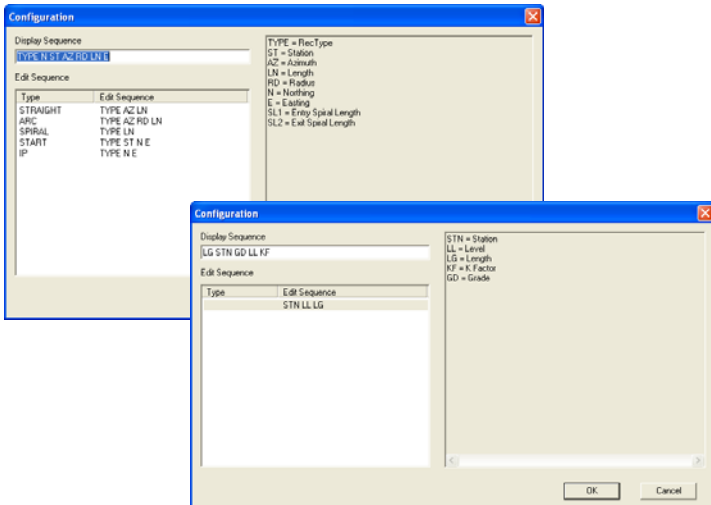


Figure 13-21. Edit Sequence Dialog Box – Alignment and Profile Tab

As data is entered in either the *Alignment* or *Profile* tab, the respective preview pane will be updated with the specified display data.

From the **Context** popup menu (Figure 13-21 on page 13-19) you can do the following:

- Cut/Copy/Paste/Delete data parameters
- Clear All records
- Insert Record a record
- Load From/Save to File
- Specify the editing sequence of any element (type) by using the **Configure** menu option (Figure 13-20 on page 13-19) on the Context popup menu (Figure 13-21) and then selecting any option under the *Edit Sequence* panel.
- Insert or Hide columns for data entry.

Enter parameters in the *Display Sequence* field from the *Configuration* dialog box (Figure 13-20 on page 13-19).

However, it is easy to drag and drop the headers to their desired location using the Hide Columns/Insert Columns commands on the Context popup menu.

Create Alignment

Use the Create Alignment command to join existing points or to create new points for the Horizontal Intersection Points (HIP) of the alignment. This defines the “frame” of the alignment. You can edit each intersection point to define the curve parameters (Figure 13-22).

The alignment entity contains all details of the alignment and if road data is attached, deleting the alignment also erases the road data.

Alignments are not layer dependent, although placing them on their own layer is recommended, especially if there is underlying line data.

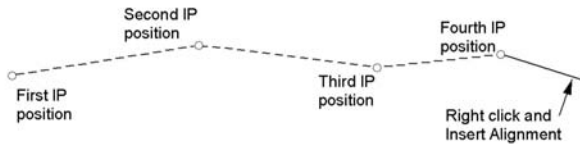


Figure 13-22. Intersection Points Selected for the Frame of the Alignment

To join existing points or create new ones, do the following:

1. Set the current layer and appropriate parameters for the new point and alignment data.
2. Click **Design** ▶ **Alignments** ▶ **Create Alignment**.
3. Click to select the first intersection point or enter the coordinates in the input bar (Figure 13-23).

Number	1	East	518.212	North	100.931
--------	---	------	---------	-------	---------

Figure 13-23. Alignment Input Bar

4. If the input mode is set to Coordinate Mode, then click the position of the next point or enter either the point number or the coordinates on the input bar.
If the input mode is set to Radiation Mode, then click the position of the next point as a bearing and distance from the start point.
5. Continue this process until all of the intersection points are defined, then select **Insert Alignment** from the right-click

shortcut menu.

The *Create Alignment* dialog box appears (Figure 13-24).

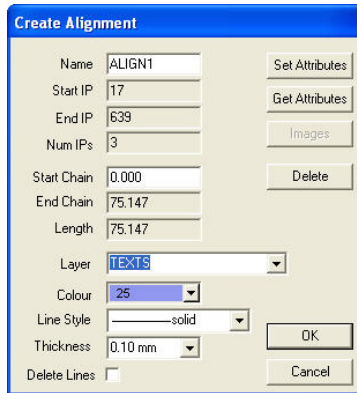


Figure 13-24. Create Alignment

6. Modify the properties as required, and then click **OK** to save the alignment. Table 13-3 describes the Create Alignment fields.

Table 13-3. Create Alignment Field Descriptions

Field Name	Field Descriptions
Name	A default name is allocated to the alignment. The first default name is ALIGN1 with subsequent alignments named as ALIGN2, ALIGN3, etc. You can edit this name, but it should not contain any spaces – use an underscore instead.
Start IP	The point number for the first HIP.
End IP	The point number for the last HIP.
Num IPs	The total number of HIPs.
Start Chain	The chainage or running distance value for the first IP of the alignment. Leave as 0.000, or enter a known running distance for the first IP.
End Chain	The chainage at the end of the alignment. This will be modified as curve parameters are applied to the IPs or when an IP is moved.
Length	The length of the alignment from the first to the last IP.
Layer	The layer on which the alignment is defined. The initial layer will be taken from the setting on the Property Bar but may be edited.
Color	The alignment color will initially be taken from the setting on the Property Bar but may be edited.

Table 13-3. Create Alignment Field Descriptions (Continued)

Field Name	Field Descriptions
Line Style	The line style of the alignment. The initial line style will be taken from the setting on the Property Bar but may be edited.
Thickness	The line thickness. The initial line thickness will be taken from the setting on the Property Bar but may be edited.
Delete Lines	Select this check box to remove lines used to define an alignment. This is not required for this option unless the selected points were already joined by lines, and the lines are no longer required for the integrity of the dataset.



You can edit the alignment once it displays in the Design View.

Create Alignment From Lines

Use the Create Alignment from Lines command to define a horizontal alignment from existing lines or strings (Figure 13-25). This procedure uses the lines or strings to define the “frame” of the alignment. The points defining these lines or strings will become intersection points (IP), which you can edit to define any curve parameter.

The underlying lines and strings may be deleted if no longer required; however, if these lines and strings represent the existing or surveyed center line, then do not delete them.

The alignment entity contains all of the alignment details. Alignments are not layer dependent, although placing them on their own layer is recommended, especially if there is underlying line data.

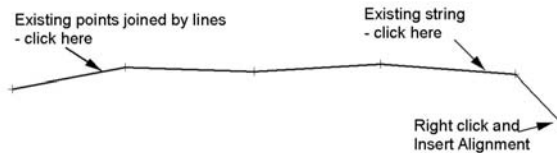


Figure 13-25. Alignment Created from Existing Lines or a String

To define a horizontal alignment from existing lines or strings, do the following (Figure 13-26) and Figure 13-27 on page 13-25.

1. Set the current layer and appropriate parameters for the new point and alignment data..
2. Click **Design ▶ Alignments ▶ Create Alignment from Lines**.
3. Select one of the following:
 - A start point, and additional points and lines to define the alignment:
 - Click to select the first existing point or enter the coordinates in the input bar.



Figure 13-26. Create Alignment from Lines Input Bar

- If the input mode is set to Coordinate Mode, then click the position of the next point or enter either the point number or the coordinates on the input bar. If the input mode is set to Radiation Mode, then click the position of the next point as a bearing and distance from the start point.
- Continue this process until all of the points are defined.
- A line close to its end point from which the alignment path continues along additional lines and arcs:
 - Click to select the next point or line, or enter either the coordinates or bearing and distance in the input bar.
 - Select the next line from the intersection point until the alignment selected is complete.
 - A string on the last segment of the string:

The string becomes highlighted and can be immediately converted to an alignment.

- Right-click and select **Insert Alignment**.

Figure 13-27. Create Alignment Dialog Box

- Modify the properties as required, and then click **OK** to save the alignment. See Table 13-3 on page 13-22 for a description of the create alignment fields.

Edit IP (Intersection Points)

Use the Edit IP command to enter details for the curve geometry at a new Horizontal Intersection Point (HIP), using the curve calculators based on the speed environment and other factors or to update and edit the existing curve geometry.

Moving the intersection point (IP) of an existing alignment does not require the recreation of arcs or spirals already defined for each IP. The existing curve data is held, although you will be prevented from moving the IP to a position where the curve parameters may no longer be applied.

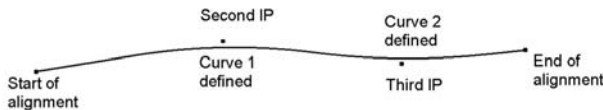


Figure 13-28. Two IPs Edited to Create Curve 1 and Curve 2

To enter details for the curve geometry, do the following:

- Click **Design** ► **Alignments** ► **Edit IP**.

2. Click on the required alignment.
The alignment and its intersection points are highlighted.
3. Click on an intersection point to edit it.
The *Edit Alignment HIP* dialog box appears (Figure 13-29).

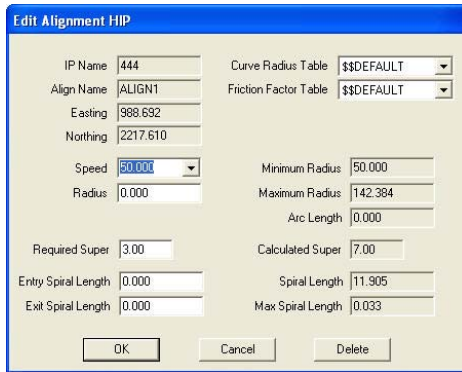


Figure 13-29. Design Options for Curve Geometry for the Alignment IP

4. On the *Edit Alignment HIP* dialog box, calculate the geometry for this curve. Table 13-4 describes the edit alignment HIP fields.

Table 13-4. Edit Alignment HIP Field Descriptions

Field Name	Field Description
IP Name	The point number of the IP.
Align Name	The alignment name.
Easting	The Easting coordinate for the IP position.
Northing	The Northing coordinate for the IP position.
Curve Radius Table	This table should be set up to follow local standards. Select a suitable curve radius table from the drop-down menu.
Friction Factor Table	This table should be set up to follow local standards. Select a suitable friction factor table from the drop-down menu.
Speed	Select a suitable speed environment for this curve from the drop-down menu. This speed is used to define the minimum radius and other values.
Minimum Radius	This is defined by the speed environment selected in the Speed field.

Table 13-4. Edit Alignment HIP Field Descriptions (Continued)

Field Name	Field Description
Maximum Radius	This shows the maximum radius that can be used for this curve within the existing geometry. This depends on the design of the previous curve and/or following curve and their definition, including the position of their IPs and tangent spiral points or tangent arc points.
Radius	Enter the required curve radius, taking into account the minimum radius displayed for the selected speed.
Arc Length	This is calculated from the radius entered, the geometry of the arc at this curve, and the tangent point positions.
Calculated Super	This is calculated from the speed environment, the radius entered, the curve geometry, and the tables specified. It is the maximum recommended super-elevation for this curve.
Required	This is initially set to the minimum super set in the preferences of the Program Setup. If this alignment and geometry is to be used for designing a road with super-elevation, update this value to match that shown in the Calculated Super field or another appropriate value. This is the percentage super-elevation value and should be entered as 4.5 for 4.5% required super-elevation.
Spiral Length	This is the recommended spiral length for designing a transitional curve, comprising a spiral–arc–spiral.
Maximum Spiral Length	This shows the maximum spiral length that can be used for this curve within the existing geometry. This depends on the design of the previous curve and/or the following curve and their definitions, including the position of their IPs and tangent spiral points or tangent arc points.
Entry Spiral Length	Enter the length of the entry spiral or leave at 0.000 for a circular curve with no transition.
Exit Spiral Length	Enter the length of the exit spiral or leave at 0.000 for a circular curve with no transition.

Move IP The coordinate of the IP may be modified by dragging the IP point to a new position. Press and hold the left **Shift** key to display the move pointer and drag the IP to a new position. The curve geometry will remain as defined for this IP, so the movement should be limited to this geometry. After moving an IP, use Edit IP to check the IP data

and make sure the defined radius and spiral lengths fit within the maximum available radius and spiral length.

To move the IP to a known coordinate, create a point at this position, and drag the IP to snap to this point with a suitable snap mode selected.

Add IP

Use the Add IP command to add on another IP at the end of the existing alignment (Figure 13-30).

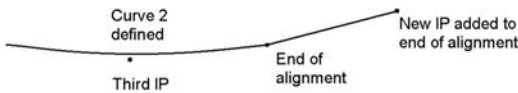


Figure 13-30. Add IP at the End of an Existing Alignment

To add on another IP at the end of the an existing alignment, do the following:

1. Set the current layer and appropriate parameters for the new point and alignment data.
2. Click **Design** ▶ **Alignments** ▶ **Add IP**.
3. Click on the required alignment.
4. Click to select an existing or new point, or enter the point number or coordinates into the input bar (Figure 13-31).



Figure 13-31. Add IP Input Bar

5. To complete the procedure, right-click and select **Cancel**.
The new IP is added to the end of the existing alignment. You can add more than one IP.
The alignment definition is now updated to allow for the new IP, changing the details on the **Edit Alignment** dialog box.

Insert IP

Use the Insert IP command to insert a new IP at the beginning of the existing alignment or in between two existing intersection points (IP).

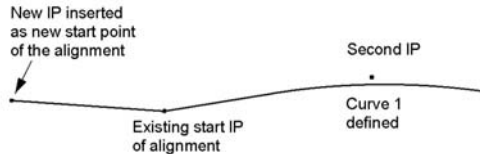


Figure 13-32. IP Inserted as the New Start Point of the Alignment

To insert a new IP at the beginning of an existing alignment, do the following:

1. Set the current layer and appropriate parameters for the new point and alignment data.
2. Click **Design** ► **Alignments** ► **Insert IP**.
3. Click on the required alignment.
4. Click on an existing IP.

The new IP is inserted before this IP.

5. Click to select an existing or new point, or enter the point number or coordinates into the input bar.
6. To complete the procedure, right-click and select **Cancel**.
The new IP is inserted into the existing alignment. You can insert more than one IP.

The alignment definition is now updated to allow for the new IP, changing the details on the **Edit Alignment** dialog box.

Delete IP

Use the Delete IP command to delete an intersection point (IP) from the definition of an existing alignment (Figure 13-33).

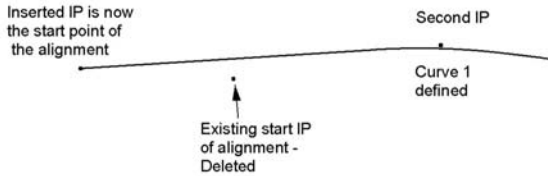


Figure 13-33. IP Original Start IP Deleted and New IP is Start of Alignment

To delete an intersection point, do the following:

1. Click **Design** ▶ **Alignments** ▶ **Delete IP**.
2. Click on the required alignment.
3. Click on an existing IP.

The new IP will be inserted before this IP.

4. Click to select an IP to remove it from the alignment definition.

The original IP number remains as a point in the database.

The alignment definition is now updated to allow for the deleted IP, changing the details on the **Edit Alignment** dialog box.

Alignment Text

Use the Alignment Text command to apply chainage or running distance labels along an existing alignment (Figure 13-34 on page 13-30). You must extract cross sectional data at specified chainage spacings before the alignment can be labelled with the chainages at these defined cross sections.

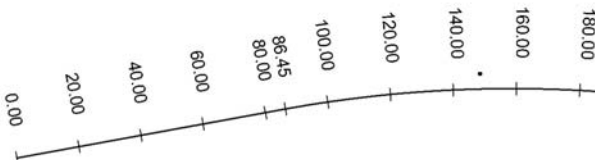


Figure 13-34. Chainage Text Annotating the Alignment

To use the Alignment Text command, do the following:

1. Click **Design** ► **Alignments** ► **Alignment Text**.
The *Alignment Text* dialog box appears (Figure 13-35).

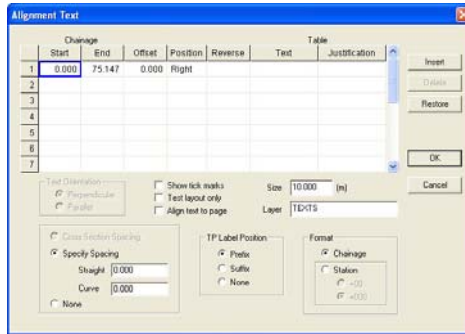


Figure 13-35. Alignment Text Dialog Box

Each row represents an alignment segment. You can fill in one row with settings for the full length of the alignment or several rows of data to apply different settings to alignment sections. This may be necessary due to the way the alignment changes direction, and the need to show the alignment text in different positions to avoid other data.

2. Enter details for labelling the alignment cross sections with their chainages, and click **OK**.

The *Additional Chainages* dialog box appears (Figure 13-36 on page 13-31).

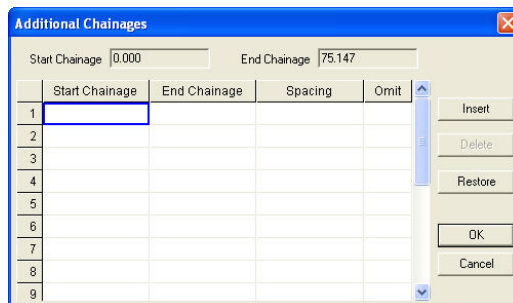


Figure 13-36. Additional Chainages

3. Enter any additional spacings or individual cross section chainages required in the report, and click **OK** to save the entered data and display the alignment text in the Survey View.
If the text is not positioned correctly, then open this option again and change the settings.
If you want to move the alignment, then delete the alignment text and recreate it in the new position.

Delete Alignment Text

Use the Delete Alignment Text command to remove the text labels written for an alignment in the **Alignment Text** command.

To remove text labels, do the following:

1. Click **Design** ▶ **Alignments** ▶ **Delete Alignment Text**.
The *Select Alignment* dialog box appears (Figure 13-37).

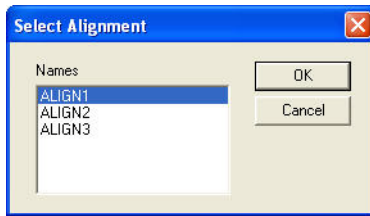


Figure 13-37. Select Alignment

2. Select an alignment to remove its text, and click **OK**.
If an alignment is selected in the Survey View before this command is opened, the **Select Alignment** dialog box does not appear, and the text is deleted.

Alignment Report

Use the Alignment Report command to create a report listing all of the alignment details. You can define spacings for the report. These may match the settings for extracted cross sections along the alignment, or you can display the details of the tangent point positions.

To create a report listing all of the alignment details, do the following:

Click **Design** ▶ **Alignments** ▶ **Alignment Report**. The *Select Alignment* dialog box appears (Figure 13-38).

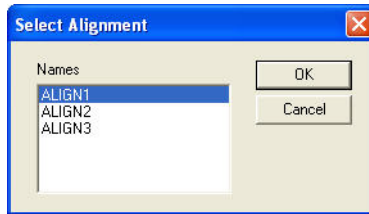


Figure 13-38. Select Alignment

3. Select an alignment, and click **OK**.

If an alignment is selected in the Survey View before this command is opened, the **Select Alignment** window does not appear.

The *Alignment List Selection* dialog box appears (Figure 13-39 on page 13-33).

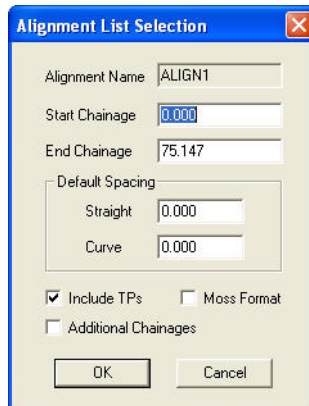


Figure 13-39. Alignment List Selection

4. Define the required parameters, and click **OK** to display the report.

The *Additional Chainages* window appears (Figure 13-40).

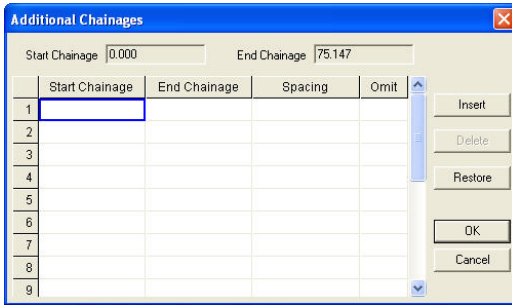


Figure 13-40. Additional Chainages

5. Enter any additional spacings or individual cross section chainages required in the report, and click **OK**.

A report listing the alignment details, based on the cross section spacings requested displays (Figure 13-41 on page 13-34).

- Alignment List Report Jul 02, 2010 11:06AM Page: 1

manual data
Alignment ALIGN1

Chainage	Easting	Northing	Radius	TanLength1	TanLength2	DefAngle	Bearing
0.000	235381.465	667124.215					53°59'33" Straight
36.491	235410.984	667145.668					53°59'33" Straight
36.491	235410.984	667145.668	110.000	54.271	54.271	52°31'14"	53°59'33" Arc
137.323	235456.280	667231.825	110.000	54.271	54.271	52°31'14"	1°28'19" Arc
137.323	235456.280	667231.825					1°28'19" Straight
270.980	235459.713	667365.438					1°28'19" Straight
270.980	235459.713	667365.438					1°28'19" Spiral1
285.280	235460.506	667379.711					6°35'35" Spiral1
285.280	235460.506	667379.711	80.000	37.787	37.787	31°37'11"	6°35'35" Arc
329.429	235477.119	667420.012	80.000	37.787	37.787	31°37'11"	38°12'45" Arc
329.429	235477.119	667420.012					38°12'45" Spiral2
343.729	235486.614	667430.697					43°20'00" Spiral2
343.729	235486.614	667430.697					43°20'00" Straight
424.170	235541.953	667489.353					43°20'00" Straight

Figure 13-41. Alignment List Report

Offset Report

Use the Offset Report command to create a report listing all of the points within a specified offset of an alignment. The list will display the chainage and offset for each of these points.

To use the Offset Report command, do the following:

1. Select the data for the report.

- Click **Design ▶ Alignments ▶ Offset Report**.
The *Offset Report* dialog box appears (Figure 13-42).

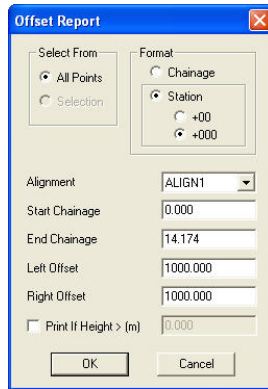


Figure 13-42. Offset Report Dialog Box

- Modify the properties as required, and then click **OK** to display the report (Figure 13-43 on page 13-35).

A report listing the required points with their offsets from the alignment and coordinate data is displayed. You can read the report on the screen, print it, or save it to a report (.rep) file.

- Offset Report Jul 02, 2010 11:06AM Page: 1

Job Name: manual data
 All points square to pegged alignment.
 Pegged Alignment: ALIGN1
 Maximum offset: 0.000

Pegged Chain	Offset	Pt Number	Easting	Northing	Height	Code
0.000	0.000	26	235381.465	667124.215		0.000
20.000	0.000	28	235394.584	667139.311		20.000
40.000	0.000	29	235407.702	667154.408		40.000
60.000	0.000	30	235420.821	667169.504		60.000
80.000	0.000	31	235433.940	667184.600		80.000
92.949	0.000	32	235442.434	667194.374		92.949
178.843	0.000	37	235469.169	667273.720		178.843
180.000	0.000	38	235469.093	667274.874		180.000
200.000	0.000	39	235467.785	667294.831		200.000

Figure 13-43. Offset Report

Convert Alignment to String

Use the Convert Alignment to String command to convert an alignment back to a simple string. Any cross section data, profile

data, or road design data will be deleted. You can save the intersection points (IP) of the alignment.



You cannot undo this conversion. Save the project before using this option, so you can close the project and end the routine without saving it if necessary.

To convert an alignment back to a simple string, do the following:

1. Click **Design ▶ Alignments ▶ Convert Alignment to String**.
2. Select the alignment you want to convert to a string.
A confirmation message may appear, depending on what is associated with the alignment (Figure 13-44).



Figure 13-44. Confirmation Message

3. In this case, click **Yes** to convert the alignment to a string with only the IPs saved as string nodes. Click **No** to convert the alignment to a string with nodes at each tangent point (TP) on the alignment. Straights are saved as line segments, arcs are saved as three-point arc segments, and spirals are converted to small line segments.

Create Center Line Points

Use the Create Center Line Points command to create points on the center line at nominated spacings on a selected layer. The code of each point will display the chainage (running distance of the point) along the alignment.

To create points on the center line at nominated spacings, do the following:

1. Click **Design** ► **Alignments** ► **Create Center Line Points**. The *Select Alignment* dialog box appears (Figure 13-45), listing all of the project alignments.

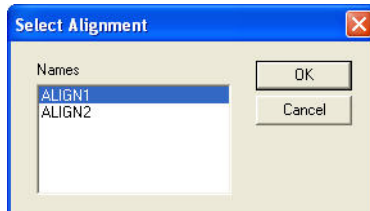


Figure 13-45. Select Alignment

2. Select an alignment, and click **OK**. The **Centre Line Points** dialog box appears (Figure 13-46 on page 13-37).



If an alignment is selected in the Design View before this command is opened, the *Select Alignment* dialog box does not display

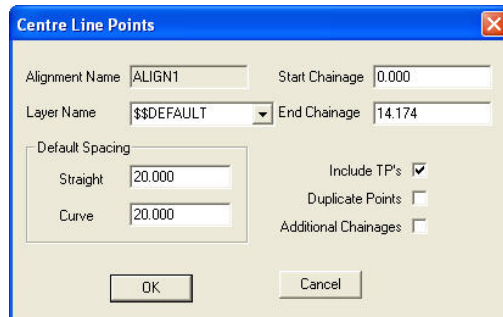


Figure 13-46. Center Line Points

3. Modify the properties as needed, and click **OK** to create the center line points.



These points are not updated when the alignment is modified. Delete the points on this layer and re-run this command.

Catch Points

Use the Catch Points command to compute catch points on a DTM surface from points on an alignment or string at a nominated grade. The points on the alignment or string must have valid elevations. This command may be used where a catch needs to be calculated from a designed line with levels to an existing surface.

To use the Catch Points command, do the following:

1. Click **Design** ▶ **Alignments** ▶ **Catch Points**.
2. Select an alignment or string.

The *Catch Points* dialog box appears (Figure 13-47 on page 13-38).

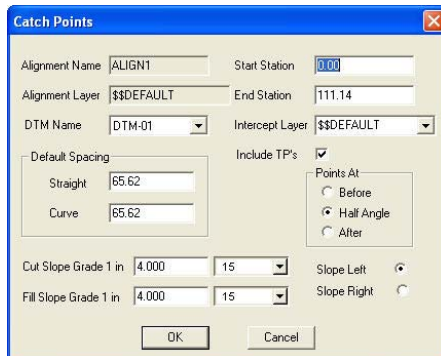


Figure 13-47. Catch Points

- Modify the properties as needed, and click **OK**. The *Preferred Chainages* dialog box appears (Figure 13-48). Table 13-5 describes the **Catch Point** fields.

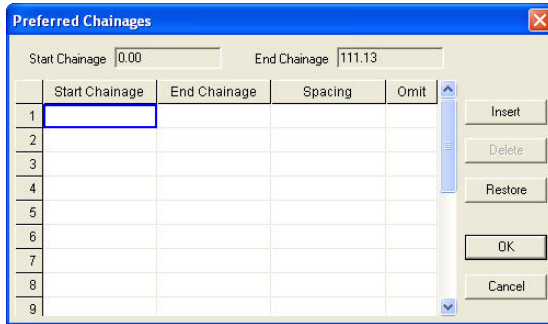


Figure 13-48. Preferred Chainages

- Enter any additional spacings or individual cross section chainages that are required, and click **OK**.

The slope points, connected by strings, are created on the defined layer. The strings are created for each segment of cut and fill with levels on the points that relate to the chosen DTM surface.

Table 13-5. Catch Points Field Descriptions

Field Name	Field Description
Alignment Name	The name of the selected string or alignment.
Alignment Layer	The layer name of the selected string or alignment.
DTM Name	Select the DTM for the computation from the drop-down menu.
Start Station	The start chainage of the selected string or alignment is displayed. You can modify this if you only want to use part of the string or alignment.
End Station	The end chainage of the selected string or alignment is displayed. You can modify this if you only want to use part of the string or alignment.
Intercept Layer	Select a layer for the new slope entities or enter a new layer name. If a new name is used, incorrect catch points may be quickly deleted.
Include TPs (Tangent Points)	Select this check box to include tangent point (TP) points from the string or alignment in the computation.

Table 13-5. Catch Points Field Descriptions (Continued)

Field Name	Field Description
Default Spacing	Enter the default spacing along Straights and Curves. If these fields are set to 0.000, then only points at the tangent point (TP) will be created if the TP option is selected.
Cut Slope Grade 1 in	Enter a grade to be used for computing cut slopes and define a color for the cut slope string and points.
Fill Slope Grade 1 in	Enter the grade to be used for computing fill slopes and define a color for the fill slope string and points.
Slope Left/Right	Select to compute the slope to the left or right of the selected string or alignment.

Create Offset Strings

Use the Create Offset Strings command to create strings parallel to a string with stations referenced to the selected alignment. You can also use this command to create additional strings parallel to the curb strings of a road design.

To use the Create Offset Strings command, do the following:

1. Click **Design** ▶ **Alignments** ▶ **Create Offset Strings**.

The *Select Alignment* dialog box appears (Figure 13-49 on page 13-40).

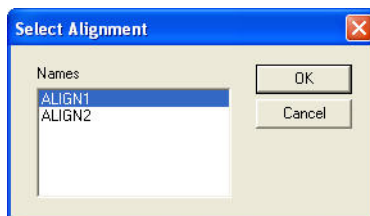


Figure 13-49. Select Alignment Window

2. Select an alignment, and then click **OK**.

The *Offset String* dialog box appears (Figure 13-50).

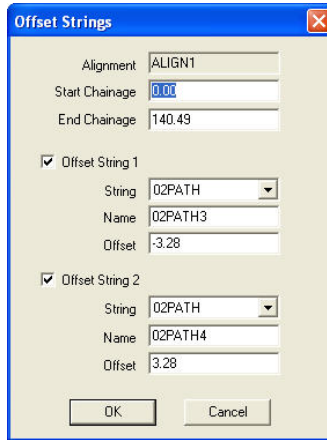



Figure 13-50. Offset String Dialog Box

3. Enter the required start and end stations on the alignment that the strings will be created between.
4. Check mark the **Offset String 1** box and choose the existing string to which the new string will be parallel.
5. Enter a **Name** for the new string.
6. Enter the **Offset** of the new string from the existing string.
7. For a second string, make sure the **Offset String 2** check box is selected and choose an existing string to which the second string will be parallel. Repeat steps 5 and 6.
8. When you have finished, click **OK**.

The offset strings appear in the display area.

Create Multi Sheet Plot Windows

Use the Create Multi Sheet Plot Windows command to create a linked drawing. The drawing is defined with a nominated plot scale, title block, and orientation. The drawing displays all of the data currently on active layers in the Design View, complete with any annotation, triangles, and contours activated in the **General Settings** (Hot Button ) of the project.

You can turn layers may on or off in the drawing. Any layer added to the project after the drawing was created may be turned on in the drawing. Any annotation that was not available at the time the drawing was created may be activated.

Several plot windows can be defined in one project, creating a linked drawing for each created plot window. You can save the drawings with unique names to indicate the content of the drawing.

To create a linked drawing, do the following:

1. Click **Design** ► **Alignments** ► **Create Multi Sheet Plot Windows**.
2. Select an alignment or string, as stated in the prompt line. The *Microsoft Windows Open* dialog box appears (Figure 13-51).

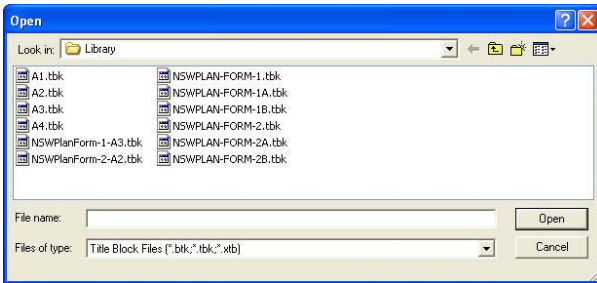


Figure 13-51. Open Dialog Box

3. Select the required title block for this drawing, and then click **Open**.
The lower left corner of the title block is displayed in the center of the screen (Figure 13-52 on page 13-42).
4. Modify the settings and position the title block around the data on the screen.

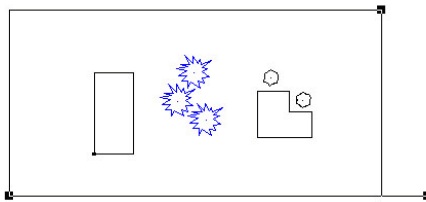


Figure 13-52. Drawing Area of the Title Block Dragged Around the Drawing Data

- The mouse can drag the plot window and snap the lower left handle to an existing point.
 - The mouse can rotate the plot window using the lower right handle or the arrow handle, or rotate it parallel to a line.
 - You can resize the plot window to find an approximate scale, using the upper right drag handle.
5. When you have defined the plot window, click **OK** on the input bar to create the new drawing and open the Drawing View (Figure 13-53).



Figure 13-53. Plot Window Input Bar

The Survey View remains open, but the Drawing View is the active window.

Additional Information

- You can define a series of plot windows in a project. These windows can overlap each other or be created at different scales. Each drawing is initially created as DrawingX, where X is the next number available. You can rename the drawing in the Drawing View.
- Once a drawing has been created, it may be reopened from the list of Drawings in the **Tools ▶ Drawing Editor** command.

Intersection

Various options on the **Design ▶ Intersection** menu may be used to build up the component curbs (stringlines) through an intersection. These curbs (curb returns) need to be defined before an intersection can be created.

Once the curbs have been created, they can be opened in Survey view by clicking **Design ▶ Curb View/Edit**.

The option on the Intersection menu are used to compute curbs (Figure 13-54). Curbs are particular strings defined through intersections between one road and another.

The various commands may be used to build up the curbs (string lines) through an intersection. Once these curbs are defined, an intersection may be created.

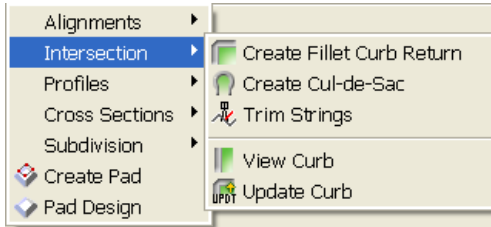


Figure 13-54. Design Menu – Intersection Menu Options

In This Section:

- “Create Fillet Curb Return” on page 13-44
- “Create Cul-de-sac” on page 13-45
- “Create Curb From String” on page 13-48
- “Create an Island Curb” on page 13-50
- “Create Intersection” on page 13-53

Create Fillet Curb Return

Use the Create Fillet Curb Return command to create a filleted arc between any two entities that are strings, lines, or arcs to define a curb

return. The end result is a graded curb line between the curb of one road and that of an intersecting road (Figure 13-55).

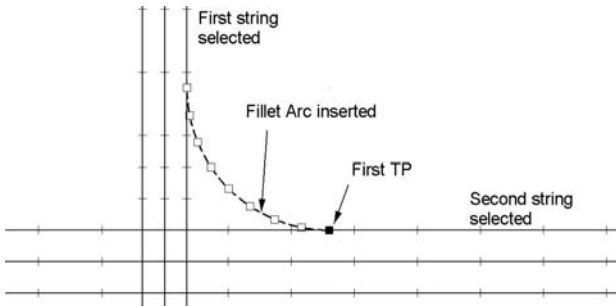


Figure 13-55. Arc Inserted Between Two Strings to Form a Curb Return

To use the Create Fillet Curb Return command, do the following:

1. Click **Design** ► **Intersection** ► **Create Fillet Curb Return**. A prompt line displays that reads: *“Select a string, line, or an arc”*.
2. Click a string, line or arc, as stated in the prompt line, that represents an existing lip or invert of a designed road curb.
3. Click another string, line, or arc that represents an existing lip or invert of another designed road curb.
An arc displays between the two selections.
4. Enter the fillet arc **Radius** in the input bar or click near the center of the fillet arc (Figure 13-56).



Figure 13-56. Radius Input Bar

5. Click on one of the yellow crosses defining the end of the fillet arc.
6. In the input bar, enter the number of **Segments** and check mark either the Divide Equally or Fix Start Point box.
7. Press **Enter**.

Create Cul-de-sac

To insert cul-de-sac parameters, do the following:

1. Click **Design** ► **Intersection** ► **Create Cul-de-sac**.

The prompt line appears.

2. Select the first curb.
3. Select the second curb.
4. Select the center point for the cul-de-sac.

The *Create a Cul-de-sac* dialog box appears (Figure 13-57).

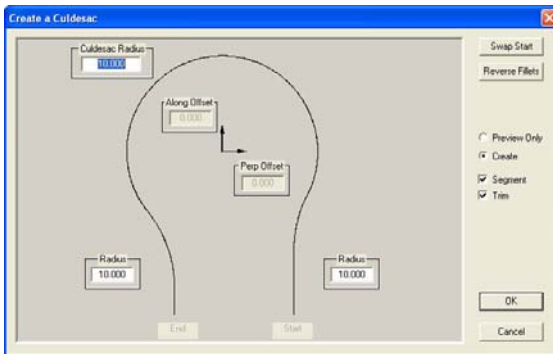


Figure 13-57. Create a Cul-de-sac Window

5. Make any necessary modifications, and then click **OK**.

The cul-de-sac displays on the screen.

Trim String

The road design generates several strings for the gutters on each side of the intersecting roads. After completing the design of the curb returns, a template is applied to those strings, creating additional overlapping strings for each curb (Figure 13-58 on page 13-47).

Use the Trim String command to locate overlapping strings and merge them with the road strings to finish the design.

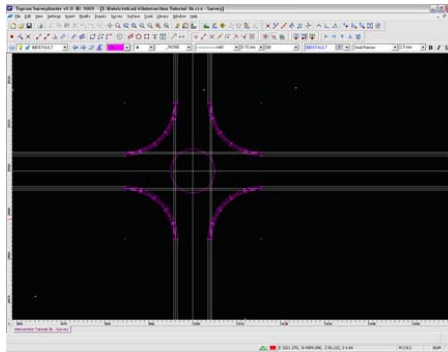


Figure 13-58. Overlapping Strings

To merge overlapping strings with road strings, do the following:

1. Drag a rectangle around the intersection, selecting the strings you want to merge (Figure 13-59).

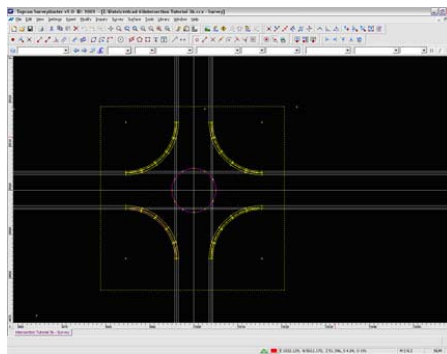


Figure 13-59. Select the Intersection

2. Click **Design** ► **Intersection** ► **Trim Strings**. A confirmation dialog box appears.
3. Click **Yes** to remove the resulting orphaned strings through the intersection.

Create Curb From String

The Create Curb from String command issued to convert a string, polygon, an arc, or a line into a curb. The curb must have starting and ending points defined. The result is a curb line that can be formed with profile data from a DTM or from the position and levels of the string notes, as long as these points have elevations.

To convert a string, polygon, an arc, or a line into a curb, do the following:

- Click **Design** ▶ **Intersection** ▶ **Create Curb from String**.

The following prompt line displays: “*Select a polyline, polygon, arc or line*”.

- Click on a string, polygon, arc, or line that represents a stringline that will become a curb. The nodes of these entities usually have elevations.

The following prompt line displays: “*Select first TP*”.

- Click on the first point on the highlighted string for the start of the curb. If the entity selected is a polygon, the result of the design line will depend on the node of the polygon selected.

The following prompt line displays: “*Select second TP*”.

- Click on the last point on the highlighted string for the end of the curb. The **Define Curb Return** dialog box appears (Figure 13-60 on page 13-49).

These two TP (tangent point) positions need not define the entire length of an existing string, if only part of a string will be used for the curb. The first TP point will become the 0.00 distance on the profile of the curb in the Curbs View.

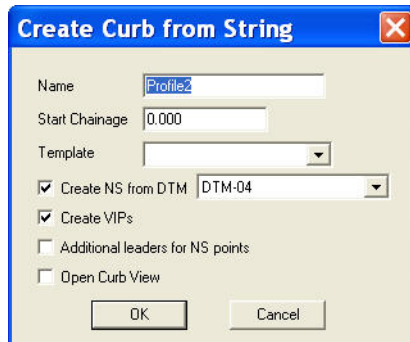


Figure 13-60. Create Curb From String

- Enter or select the following parameters:
 - Name: A default name displays (starting with Profile1). You can enter a new name. No spaces are allowed in the curb name.
 - Start Chainage: Default is 0.00 but can be modified if the running distance on the profile plot is starting at a negative distance or alternative positive value.
 - Template: Select a desired surface for the curb.
 - Create NS from DTM: Check mark this box to extract the natural surface of the curb profile from the DTM selected from the drop-down box.
 - The initial DTM is the current one.
 - This box should not be check marked if you want no natural surface to display in the Curbs view.
 - Create VIPs: check mark this box to define the “design surface” that displays in the Curbs View as a best-fit curve through the nodes. If the note points on the string have elevations, these may be defined as the VIPs of the Design surface on the profile.
 - Additional leaders for NS points: Check mark this box to create additional leader lines for each change in level on the natural surface.
 - Open Curb View: Check mark this box to display the Curb View.

- Click **OK** to save the curb parameters.
- Click **Cancel** to quit the routine.

The computed curb will have a profile defined along the curb from the surface chosen (DTM or VIPs) and the design for the curb will be a best-fit curve with 4VIPs.

The curb will remain highlighted and can be viewed/updated in the Curbs view by clicking **Design ▶ Intersection ▶ View Curb ▶ Update View**.

The curb design can be modified in the Curbs View and then saved back into the Survey View.

Create an Island Curb

Use the Create an Island Curb command to create a curb around the extents of a polygon or of a closed shape. One node of the polygon or shape will be the starting point of the curb. The polygon or shape will usually represent a traffic island. A circular roundabout can be defined as a circle.

To create an island curb, do the following:

1. Click **Design ▶ Intersection ▶ Create an Island Curb**.
The prompt line displays: “Select a polygon or a segment of an enclosed area”.
2. Click a polygon or a segment of lines and arcs that make up a closed shape or a circle.
The shape could be defined by using Locate House to insert a pre-designed traffic island. If the shape is a circle, the prompt line displays: “Locate a reference point on the circle”.
3. Click the one point on the circle:
The prompt line displays: “*Enter the number of segments (2 or greater)*”.
4. Define the number of segments required around the traffic island, then click in the view to accept the settings and continue.
The prompt line displays: “*Select start point for the curb*”.

- Click on one point that will be the starting point of the curb and the 0,00 distance on the profile of the curb in Curb View. The *Curb Point Level* dialog box appears (Figure 13-61).

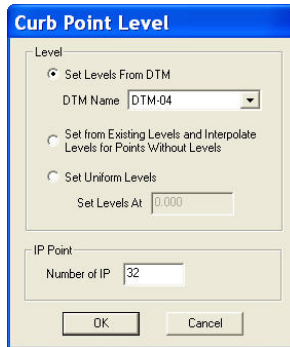


Figure 13-61. The Curb Point Level Parameters

- On the *Curb Point Level* dialog box, enter or select the following parameters.
 - Set the one of the following parameters for the curb point level:
 - **Set Levels From DTM:** If this option is selected, choose a DTM name from the drop-down list.
 - **Set from Existing Levels and Interpolate Levels for Points Without Levels:** If this option is selected existing levels and interpolated levels will be used.
 - **Set Uniform Levels:** If this option is selected, enter a required level number.
 - IP Point: Enter the number of IPs to display on the main screen.
- Click **OK**. The *Create Island Curb from polygon* dialog box appears (Figure 13-62 on page 13-52).

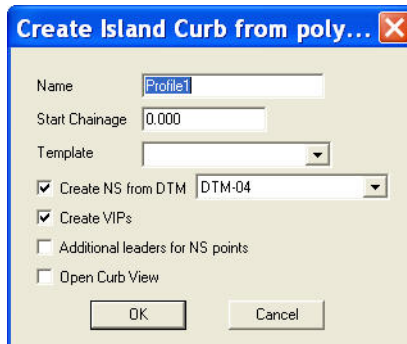


Figure 13-62. Create a Curb Island From a Polyline

8. On the *Create Island from polyline* dialog box, enter or select the following parameters:
 - Curb Name: A default name displays (starting with Profile1). You can enter a new name. No spaces are allowed in the curb name.
 - Start Chainage: Leave the start chainage at 0.000.
 - Template: Select a template from the drop-down list to define the surface at the island.
 - Create NS from DTM: If this box is check marked, the natural surface of the curb profile will be read from the DTM selected from the drop-down box.
 NOTE: The initial DTM is the current DTM. Uncheck mark this option if the “natural surface (NS)” that displays in Curb View is taken from the elevation points on the entity nodes.
 - Create VIPs: Check mark this box to define the surface that displays in the Curbs/Intersection View from the elevation points on the entity nodes.
 - Additional leaders for NS points: check mark this box to create additional leader lines for each change in level on the natural surface.
 - Open Curb View: Check mark this box to open the Curb View to check the results.
9. Click **OK** to save the Curb.

- Click **Cancel** to quit the routine.

The computed curb will have a profile defined along the curb from the surface chosen (DTM or VIPs) from the point selected. This curb may now be used with the other curbs in the Intersection Design.

Create Intersection

The Create Intersection command will create an intersection from selected curbs and string profiles. The Intersection can be opened in Intersection View for comprehensive intersection design to grade all stringlines and to create a suitable surface model.



Be sure that all unneeded layers are turned OFF. An intersection should be created only from the Curb Returns and the CL strings of the road. You may also need other strings between the CL and curb returns if these string impact on the final surface through the intersection.



Do not include the “curb” strings from the roads outside the intersection. They have already been designed in Roads View.

To create an intersection, do the following:

- Click **Design ▶ Intersection ▶ Create Intersection**.

The “*Create a rectangle to define the intersection – Select first point*” prompt displays at the bottom of the screen.

- Left-click with the mouse to define the bottom-left corner of the rectangle that will define the extents of the Intersection.

The “*Create a rectangle to define the intersection – Select second point*” prompt displays at the bottom of the screen.

- Left-click with the mouse to define the top-right corner of the rectangle that will define the extents of the Intersection.

The *Create Intersection* dialog box appears (Figure 13-63).

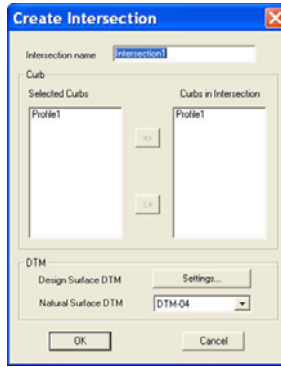


Figure 13-63. Create Intersection Parameters

Table 13-6 describes the field options for creating an intersection

Table 13-6. Create Intersection Field Options

Field Name	Field Description
Intersection Name	A default name displays (Intersection1). You can change the name to something else.
Curb	
Selected Curbs	To remove a curb, from the Intersection, highlight the Curb name and click on the left arrow key.
Curbs in Intersection	To add a curb into the Intersection, highlight the Curb name and click on the right arrow key.
DTM	
Design Surface DTM	Press the Settings button to define appropriate settings for the new Design Surface DTM. NOTE: These settings can be made now or later in Intersection Design.
Natural Surface DTM	Select a natural surface DTM from the drop-down list.
OK	Press to create the Intersection.
Cancel	Press to quit the routine.

To review a previously created intersection, click **Design ▶ Intersection ▶ View Intersection**. You can open, rename, or delete the highlighted intersection from the *Intersections* dialog box.

Design Profiles

Profiles represent the existing, design, and final configurations of a surface. Usually, a design profile sheet shows the existing ground profile before any construction has begun and the final profile after all construction is completed (Figure 13-64).

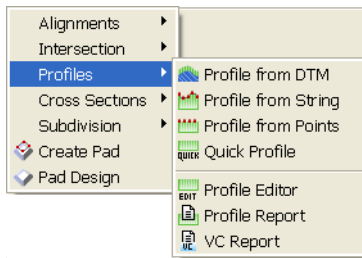


Figure 13-64. Design Menu – Profile Options

In This Section:

- “Profile From DTM”
- “Profile From String” on page 13-57
- “Profile From Points” on page 13-58
- “Quick Profiles” on page 13-60
- “Profile Editor” on page 13-60
- “Profile Report” on page 13-63
- “Quick Profile” on page 13-63

Profile From DTM

Use the Profile from DTM command to create profiles from an existing DTM and a string. The profile created from a string can be edited to design a final configuration, including vertical curves, if the Create VIPs option is selected. The surface does not need to be displayed in order to create the profile from the DTM, however, a surface must be contained in the project.

To create a profile from a DTM, do the following:

1. Click **Design** ▶ **Profiles** ▶ **Profile from DTM**.

2. Select a string, as stated in the prompt line on the lower left of the window. The points that comprise the string can have elevations, but it is not necessary to create the profile.

The **Create Profile from DTM** dialog box appears (Figure 13-65).

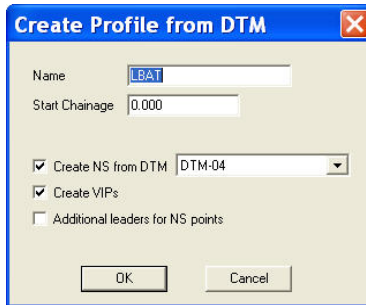


Figure 13-65. Create Profile from DTM

3. Modify the properties as necessary, and then click **OK**. For a description of the fields, see Table 13-7 on page 13-57.

The profile is saved within the unified project file and can be viewed in the Profile Editor (**Design ▶ Profiles ▶ Profile Editor**). The **DTM** profile dialog box displays (Figure 13-66 on page 13-57).

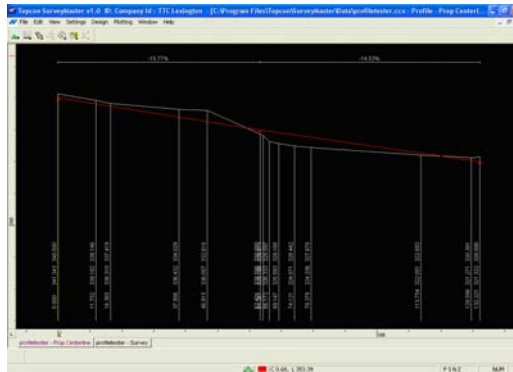


Figure 13-66. Profile from DTM with All Options Selected

Table 13-7. Create Profile from DTM Field Descriptions

Field Name	Field Description
Name	The name of the profile to be created.
Start Chainage	Determines the profile starting station.
Create NS from DTM	Creates a profile from a specified DTM. When creating VIPs from the string, the DTM will reflect the natural surface (NS) profile. To specify a DTM, select it from the drop-down menu. If this check box is not selected, then only the selected string will be used to create the profile.
Create VIPs	Creates vertical intersection points (VIP) from the selected string. Usually, the string would represent a design grade. String VIPs can be added, deleted, and modified in the profile editor. If this check box is not selected, then there will not be a profile created for the highlighted string. When using this option, the color of the selected string will be used to represent the string profile in the Profile Editor.
Additional Leaders for NS Points	Draws leaders to each break point encountered in the DTM along the profile alignment.

Profile From String

Use the Profile from String command to create a profile from points along a string.

To create a profile from a string, do the following:

1. Click **Design** ▶ **Profiles** ▶ **Profile from String**.
2. Select a string, as stated in the prompt line on the lower left of the dialog box. The string points must have elevations.
The *Profile from String* dialog box appears (Figure 13-67).



Figure 13-67. Profile From String Dialog Box

3. Enter the name of the profile you want to create.
4. Select the **Open Profile View** check box to display the profile in the Profile Editor, and then click **OK**.

The profile is saved within the unified project file and can be viewed in the Profile Editor (**Design** ▶ **Profiles** ▶ **Profile Editor**).

Profile From Points

Use the Profile from Points command to create a profile from points along a string as well as points within a specified offset distance from the selected string.

To use the Profile from Points command, do the following:

1. Click **Design** ▶ **Profiles** ▶ **Profile from Points**.
2. Select a string, as stated in the prompt line on the lower left of the window. The points that comprise the string can have elevations, but it is not necessary to create the profile. String points can be omitted from the profile if desired.

The *Profile From Points* dialog box appears (Figure 13-68 on page 13-59).

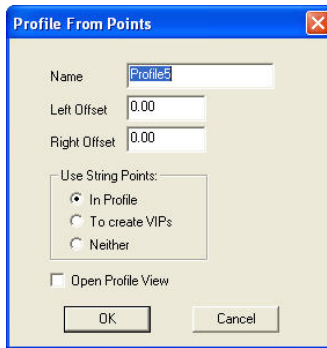


Figure 13-68. Profile From Points

3. Modify the properties as necessary, and then click **OK**. For a description of the fields, see Table 13-8 below.

The profile is saved within the unified project file and can be viewed in the Profile Editor (**Design ► Profiles ► Profile Editor**).

Table 13-8. Profile From Points Field Descriptions

Field Name	Field Description
Name	The name of the profile to be created.
Left Offset	Perpendicular distance to the left for points from the selected string. Points within this distance are included in the profile.
Right Offset	Perpendicular distance to the right for points from the selected string. Points within this distance are included in the profile.
In Profile	Select this option to include the string points in the profile. String points require elevation when using this option.
To Create VIPs	Select this option to create vertical intersection points (VIP) from the selected string points. Typically, the string represents a design grade. String VIPs can be added, deleted, and modified in the profile editor. When using this option, the selected string color is used to represent the string profile in the profile editor.
Neither	When checked, string points will not be used in the profile.
Open Profile View	Displays the profile in the Profile Editor after you click OK . You can edit the profile in the Profile Editor.

Quick Profiles

Use the Quick Profile command (Figure 13-69 on page 13-60) to create a profile from selecting a line or alignment. A surface (DTM) is required for this routine. If a surface is not present in the project, you may get an error message.

To create a quick profile, do the following:

1. Click **Design** ▶ **Profiles** ▶ **Quick Profile**.
2. Select the first point or line, as stated in the prompt line on the lower left side of the window.
3. Select the next point or line.
4. Continue this process until the alignment is defined.
5. When completed or after a line has been selected, right-click and choose **View Profile** from the shortcut menu.

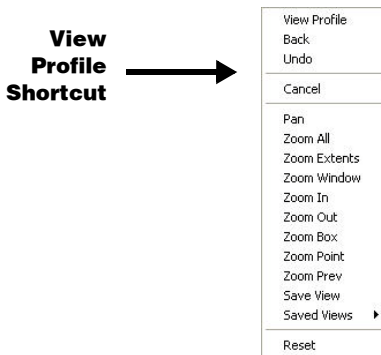


Figure 13-69. Right-Click Shortcut Menu

The profile displays in the Profile Editor. This provides a quick analysis of a surface for planning purposes. For this reason, you cannot edit the profile, but you can plot the profile if necessary.

Profile Editor

Use the Profile Editor command to modify the intersection points (IP) and curve data relating to the vertical profile design lines. You can also create plots from the Plotting menu (Figure 13-70 on page 13-61).

To modify the IP and curve data, do the following

1. Click **Design** ► **Profiles** ► **Profile Editor**.

The *profiles* dialog box appears (Figure 13-71).

2. Select a profile, and click **Open**.

The profile is displayed in the profile editor. Use the **Design** menus to add, move, and modify IPs.

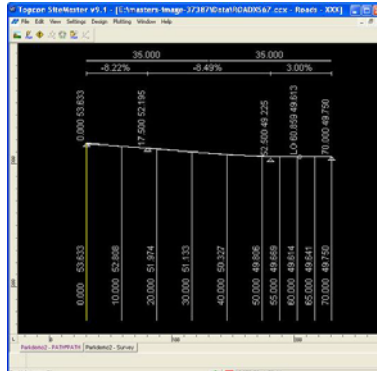



Figure 13-70. Profile Editor

3. Click the **Hot Button**  (Figure 13-70 on page 13-61) to open the *Profile* dialog box (Figure 13-71). This window lets you control the display of data on the screen.

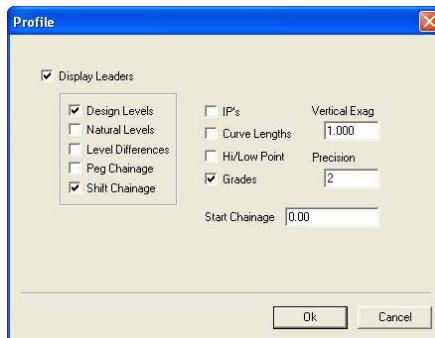


Figure 13-71. Profile Dialog Box

- Modify the properties as necessary, and then click **OK** to close the window and refresh the screen display. For field descriptions, see Table 13-9 below.

Table 13-9. Profile Window Field Descriptions

Field Name	Field Description
Display Leaders	Displays the vertical leader lines on the screen.
Design Levels	Displays the levels for the vertical profile design on each leader line.
Natural Levels	Displays the levels for the natural surface on each leader line.
Level Differences	Display the difference between the levels on the two surfaces
Peg Chainage	Display the chainage or running distance.
Shift Chainage	Displays the chainage or running distance.
IPs	Displays the level of the IP point for the profile design. The chainage and level is displayed vertically above the triangular symbol used to define the IP position.
Curve Lengths	Displays the length of the vertical curves for each IP point of the profile design. The curve length is displayed at the top of the screen above a horizontal line that shows the position and length of the curve.
Hi/Low Point	Displays the high and low points on the formation levels of the vertical profile design. The chainage and level is displayed vertically above the high or low point and marked HI for high point and LO for low point.
Grades	Displays the grade of the designed line between two adjacent IP points. the grade is displayed as a positive or negative percentage below the curve length data at the top of the screen and above a horizontal line that shows where the grades change at each IP point. The grade is measured from left to right, from the lower chainage or distance position to the higher chainage or distance position.
Vertical Exag.	This value defaults to 1.000. Enter an increased value to change the vertical scale as a ratio to the horizontal scale in the display on the screen. For example, set the value to 2.000, so the vertical scale is twice the horizontal scale.
Precision	This value defaults to 2 and sets the number of decimal places in the data displayed on the screen.

Profile Report

Use the Profile Report command, **Design ▶ Profiles ▶ Profile Report**, to produce a report of the station and elevation of the break points along the profile line.

Quick Profile

Use the VC Report command to calculate levels for the different strings involved in a design, such as the addition of a curb and channel to an existing road or widening a lane of a road using the existing pavement grading.

To generate a VC report, do the following:

1. Click **Design ▶ Profiles ▶ Quick Profile**.

The *VC Details* dialog box appears (Figure 13-72).

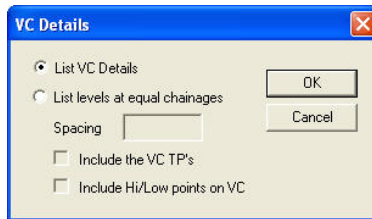


Figure 13-72. VC Details Window

2. Select your preferences, and then click **OK**.

The *VC Report* appears (Figure 13-73).

SiteMaster - Profile Report Jul 02, 2010 11:06AM Page: 1

Parkdemo2.ccx
Profile Name: NAIL

Station	I.P. Level	VC Length	Grade
0+00	343.205	0.000	78.24%
1+21	437.994	30.000	-484.83%
1+41	343.345	0.000	-12.71%
2+61	328.083	0.000	

Figure 13-73. VC Report

Cross Sections Menu

The various methods used to extract cross sections to create the road are discussed in this section (Figure 13-74).

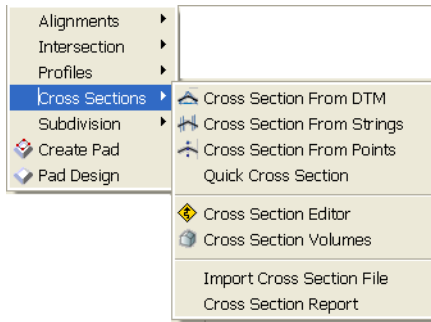


Figure 13-74. Design Menu – Cross Sections Options

In This Section:

- “Cross Section From DTM”
- “Cross Sections From Strings” on page 13-67
- “Cross Sections From Points” on page 13-68
- “Quick Cross Section” on page 13-70
- “Cross Section Editor” on page 13-74
- “Cross Section Volume” on page 13-76
- “Import Cross Section Files” on page 13-76
- “Cross Section Report” on page 13-77

Cross Section From DTM

The Cross Section From DTM command allows you to extract cross sections for a selected alignment from a triangular mesh, for use in road design and plotting. This mesh represents a surface, which is usually the existing or natural surface. Points are added to the cross section where the cross section intersects triangle sides, providing offset and level data. Cross sections are extracted from the current DTM (triangular mesh) as set in the **Hot Button**.

To extract cross sections for an alignment, do the following:

- Click **Design** ▶ **Cross Sections** ▶ **Cross Section From DTM**.
- The *Select Assignment* dialog box appears (Figure 13-75).
- Select an alignment file and press **New**.
- If the selected alignment is not active MAGNET Office displays a confirmation box that prompts you to make the alignment active.
- Press **Yes** to continue or **No** to cancel.

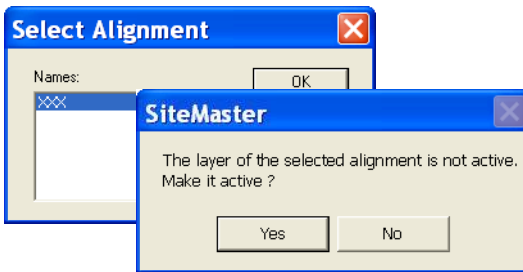


Figure 13-75. Select Alignment Dialog Box

- If you press Yes, the *Extract Cross Sections from DTM* dialog box appears (Figure 13-76 on page 13-66).

On the *Extract Cross Sections from DTM* dialog box (Figure 13-76 on page 13-66), do the following:

- Surface Name – the first extracted surface is called NATURAL. Other surfaces can be used for more data; these should be given a suitable name. However, do not use DESIGN, BOXING, or a SUBGRADE name.
- Start/End chainage – these field will automatically contain the start and end chainages of the selected alignment.
- Left/Right Offsets – these values should be greater than the expected finished batter offsets. Any data not used will be ignored.

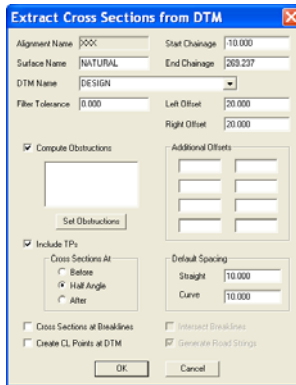


Figure 13-76. Extract Cross Sections From the DTM

- Press OK to display the *Additional Chainages* table (Figure 13-77).

On the Additional Chainages table, the user can select different cross section intervals at different chainage ranges. Individual chainages can also be included here. Chainages can be omitted by checking the OMIT column.

Single cross sections can be added or omitted simply by entering the chainage in the Start Chainage column and by leaving the End Chainage column blank. This cross section data will be stored in a file and the user can then enter the Roads module.

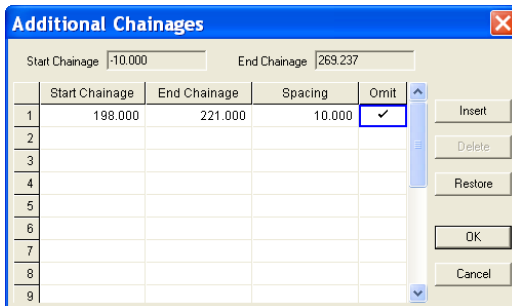


Figure 13-77. Start Additional Chainages

Cross Sections From Strings

Use the Cross Sections From Strings command to extract cross sections for a selected alignment from stringline data. The stringlines are created using the *String* option from the **Insert Menu** and are required to have heights assigned to their end points. If your lines are not strings, they will need to be breaklines to be used with this option. The cross section points are created where they intersect each string, interpolating an offset and a level. Using this method, the cross sections can be created without the job being triangulated.

- Click **Design** ▶ **Cross Sections** ▶ **Cross Sections From String**. The Extract *Cross Sections from Strings* dialog box appears ().
- Press OK after parameters have been entered for the string. The Additional Chainages table appears

For further details on parameters in Figure 13-78, see “Cross Section From DTM” on page 13-64.

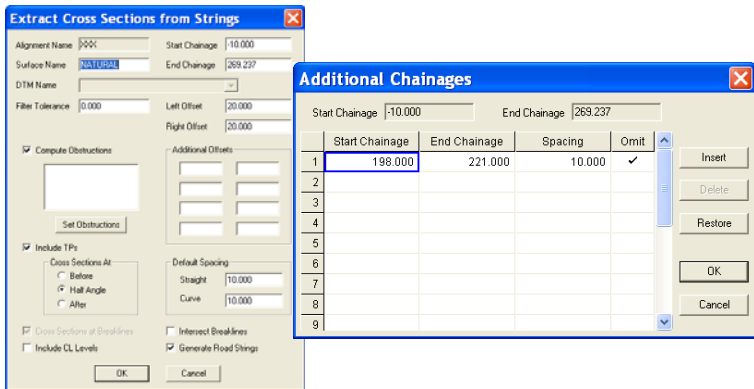


Figure 13-78. Extract Cross Sections From Strings

Only data on active layers will be used in this routine. If cross sections are only required at the surveyed pin or peg points on the alignment, do not use the Additional Chainages table.

All points extracted using Cross Section From Strings will have a code, based on the string or the point code name.



Remember to turn Off layers that hold data that should not be in the cross sections, e.g., underground services ,culverts, etc.

Cross Sections From Points

Use the Cross Sections from Points command to create cross sections for road design from points that have been observed in cross section format. Use this option when the centerline has been marked in the field (e.g., pin or peg point) and the surveyor has observed data in the form of cross sections from this centerline and reduced it into MAGNET Office. Using this method, the cross sections can be created without the triangulating the job. No interpolation occurs in this method; the option simply extracts the points on layers turned On that are within the user-defined envelope. The detail picked up in the field can be manipulated so that it is shifted on to a design centerline and additional chainages can be selected.

An observation should be made at each marked point on the centerline and the point should be coded CHMARK (555). Cross sections will be formed at each point with this code. It is essential that a height be observed for each of these points.

The tolerance parameters define an envelope around each point coded CHMARK (555). Any point that lies within this envelope will be assigned to the cross section at that chainage.

Points can only be assigned to one cross section.

To create cross sections for road design, do the following:

Click **Design** ▶ **Cross Sections** ▶ **Cross Sections From Points**.

The *Cross Sections From XYZ* dialog box appears (Figure 13-79 on page 13-69).

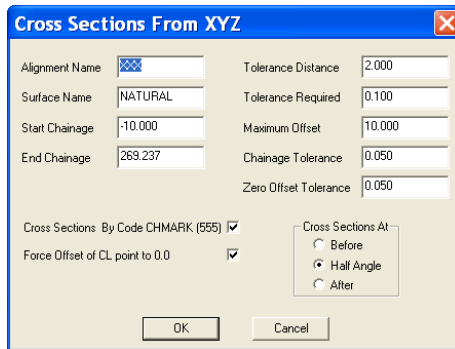


Figure 13-79. Extract Cross Sections From Points (XYZ)

Table 13-10 describes the Cross Sections From XYZ fields.

Table 13-10. Cross Sections From Points Field Descriptions

Field Name	Field Description
Alignment Name	User Defined name using numerical or alpha-numeric characters.
Surface Name	Name of the surface being crated. First surface will be "Natural".
Start/End Chainage	Start chainage and end chainage at where you want cross sections to be created.
Tolerance Distance	Enter the required distance.
Tolerance Required	Enter the distance from the centerline at which the envelope will begin to widen for containment of outer points.
Maximum Offset	Enter the maximum offset at which to search for points.
Chainage Tolerance	Enter the tolerance of centerline observations from the marked points.
Zero Offset Tolerance	Enter a tolerance here so that marked centerline points will be assigned the preferred chainage (compensates for slight observation errors) rather than the actual chainage.
Cross Sections By Code CHMARK (555)	Check mark this box so that observed points with the CHMARK (555) code will be forced onto the design alignment's centerline.
Forced Offset of CL point to 0.0	Check mark this box to force a 0.0 offset of the centerline.
Cross Sections At	Select either Before, Half Angle, or After

Table 13-10. Cross Sections From Points Field Descriptions

Field Name	Field Description
OK	Press to save.
Cancel	Press to quit the routine.



A file of cross sections will be created and the user can then enter the Road module.



All points extracted using the Cross Section From Points command will have the code of the Point as recorded in the Survey database.

Quick Cross Section


The Quick Cross Section command allows you to create cross sections for viewing without having to create an alignment and to extract the cross sections. The cross sections are defined by specifying spacing along a centerline that is defined by two or more points in sequence, by a line or series of lines, or by a string. An existing alignment may also be used.

The Cross Section View displays the cross sections at a specified spacing with specified offsets to the left and to the right of the defined centerline. A centerline profile also displays.



Before creating the profile, check that a DTM surface is correctly defined and that the required surface is set as the Current Surface.

To create a quick cross section, do the following:

- Click **Design** ▶ **Cross Sections** ▶ **Quick Cross Section** or click the  icon.

The current non-active DTM displays.

The prompt line, “*Locate the first point or line*”, displays.

- Select a start point, an existing line close to the end of the line, or an existing string.

If the entity selected is a line (a series of connected lines or strings), the sequence will be highlighted.

If a point is selected, the prompt line, “Locate next point” displays.

- Select a second point, followed by subsequent points, if required. The points do not need to exist. They will be created at the mouse cursor position.
- Right-click the mouse to display the context menu and select View Section. The *Display Settings* dialog box appears (Figure 13-80).

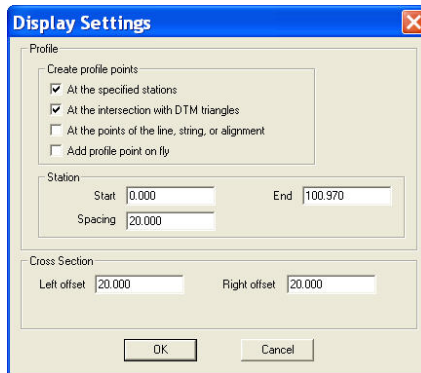


Figure 13-80. Display Setting Dialog Box

Table 13-11 describes the Display Settings field options.

Table 13-11. Display Settings Field Descriptions

Field Name	Field Descriptions
Profile	
Create profile points at specified stations	Check mark this box to display sections at the stations defined below by the start and end station values using intermediate spacing.
At the intersection with DTM triangles	Check mark this box to display sections at every point where the centerline intersects with a triangle edge.

Table 13-11. Display Settings Field Descriptions

Field Name	Field Descriptions
At the points of the line, string, or alignment	Check mark this box to display sections at every point clicked when defining the centerline or at each point used to define the line, string, or alignment entities.
Add profile point on fly	Check mark this box to enable the add a cross section at a new position in the cross section view.
Station	
Start	Enter the parameter for the starting position for the station. Sets to 0.000 station position.
Spacing	Enter the required spacing between stations.
End	Enter the final station position of the selected centerline.
Cross Section	
Left offset	Enter an offset at the left of the centerline.
Right offset	Enter an offset at the right of the centerline.
OK	Press to create the cross section view. The Cross Section View pane appears (Figure 13-81 on page 13-73).
Cancel	Press to exit (quit the routine).

The Cross Section View pane (Figure 13-81 on page 13-73) consists of three panes of data:

- Top left pane – shows the data in the Survey View.
- Top right pane – shows the Cross Section for the station selected, with the offset elevation values for each leg of the section.
- Lower pane – shows the profile along the centerline, with the station chainage and elevation at the centerline for each cross section.

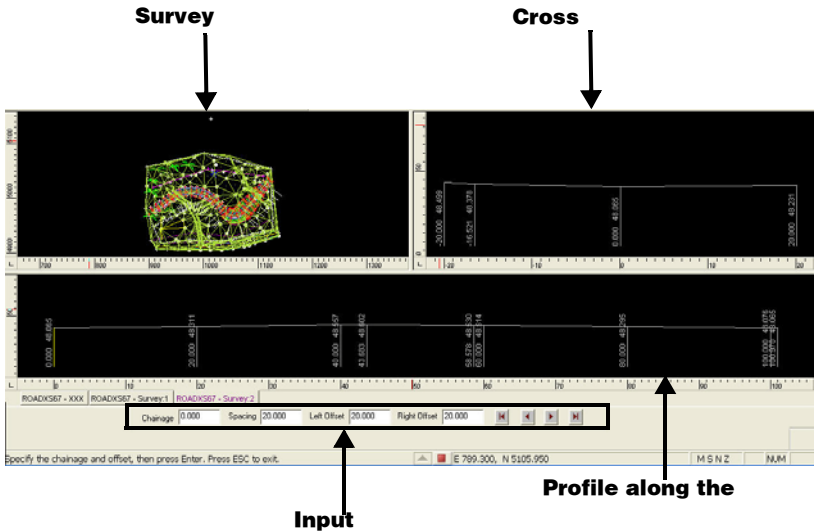


Figure 13-81. Cross Section View Display Panes

Each pane has a context menu that displays zoom options; alternatively, use the scroll wheel of the mouse. The Cross Section and Profile panes have options to exaggerate the vertical scale of the display.

The prompt line, “*Specify the chainage and offset, then press Enter. Press ESC to exit*”, appears at the bottom of the screen (Figure 13-81).

The input bar (Figure 13-81) displays the current station position and the spacing and offset previously entered, together with arrow buttons to move through the sections.

The cross section that displays relates to the section at the yellow highlighted leader line on the profile and the chainage shown on the input bar (Figure 13-82).

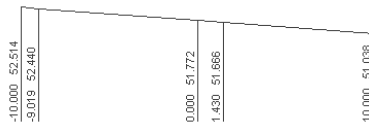


Figure 13-82. Cross Section on Display

Modify the chainage and offset as required or move through the sections at the nominated spacings using the arrow keys (Figure 13-83).

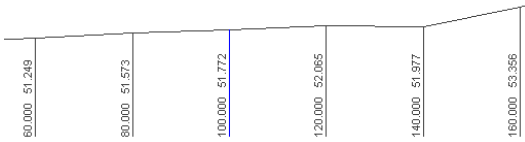


Figure 13-83. Modify Parameters

- Change the offset values to decrease or increase the width of the section shown.
- Alternatively, click on a leader line in the Profile pane (bottom) (Figure 13-81 on page 13-73) to view a particular section.
- Press the ESC key to exit when the viewing is complete.



The cross sections created by this option is not saved.




To create and define an alignment, define a string and use either *Cross Sections From DTM*, *Cross Sections From String*, or *Cross Sections From Points* from the **Cross Sections** sub-menu options under the **Design** menu.

Cross Section Editor

To edit cross sections, an alignment should have been defined and cross sections should have been extracted from a suitable surface using the “*Cross Sections From...*” options on the **Design /Cross Sections** menu.

You can open the Roads View using the Cross Section Editor command or the “Road View/Edit” menu option under the Design menu.

To open the Roads View, do the following:

- Click **Design ▶ Cross Section ▶ Cross Section Editor** or press the icon . The **Roads** dialog box appears (Figure 13-84) and displays the available Roads data listed by alignment name.
- Highlight an alignment name and press **Open**.
The Roads View opens to display the usual three panes of data.

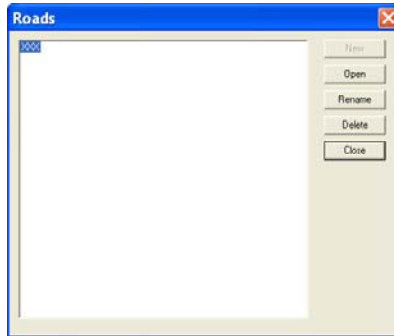


Figure 13-84. Roads View Menu Option

- To rename a filename, highlight it and press **Rename**.
- To delete a filename, highlight it and press **Delete**.

The options available on the various menus will depend on the module in use.

- MAGNET Office has options on the CrossSection menu and the Profile menu. Plotting functions are available from the plotting menu. All other options will be grayed out (unavailable). Check the Cross Section Editor options in the MAGNET Office Help manual.
- MAGNET Office has all the Road Design options available for use. Help on the use of the Design options in the Roads View are available from the Roads Help manual.

Cross Section Volume

To create a Cross Section Volume report, do the following:

- Click **Design** ▶ **Cross Section** ▶ **Cross Section Volume**. The *Cross Section Volumes* dialog box appears (Figure 13-85).
- On the *Cross Section Volumes* dialog box, enter or select the desired parameters.
- Press **OK** to generate the **Cross Section Volumes Report**.

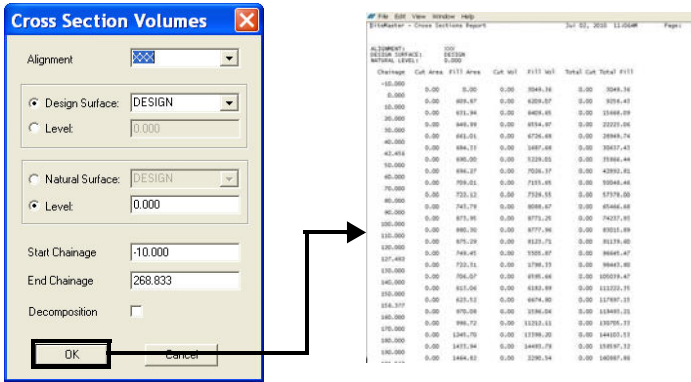


Figure 13-85. Cross Section Volumes Report

Import Cross Section Files

The following file types can be imported into MAGNET Office.

MAGNET Office Cross Section Import File Formats	
Cross Section Files (*.xxx)	GENIO Files (*.gen)
Standard Survey File (*.rd)	CLIP File (*.trv)
ISPOL File (*.sc1)	InRoads File (*.soe)

Cross Section Report

The Cross Section Report command allows you to produce a report showing a cross section listing for the current surface.

To create a cross section report, do the following:

- Click **Design** ► **Cross Sections** ► **Cross Section Report**. The *List Cross Sections* dialog box appears (Figure 13-86).
- On the *List Cross Sections* dialog box, select the Surface Name from the drop-down list.
- Click **OK** to generate the **Cross Sections Report** (Figure 13-86).

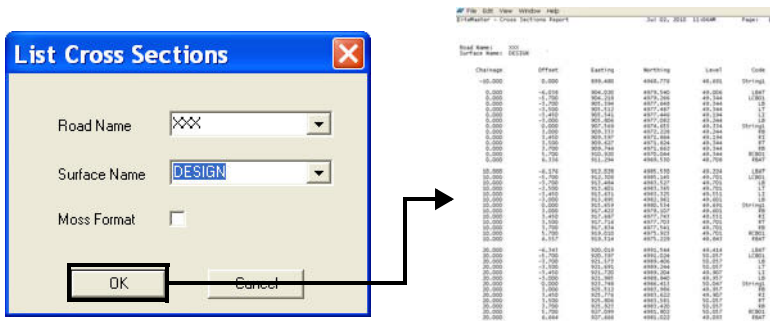


Figure 13-86. Cross Section Report

This report can be read onscreen, printed out, or saved to a report (.rep) file.

Subdivision

The Subdivision menu contains numerous options that enable you to create “lots” using minimum area calculations, define closed areas as lots, and list the lot details (Figure 13-87 on page 13-78). You can also annotate the lots to suit local planning requirements.

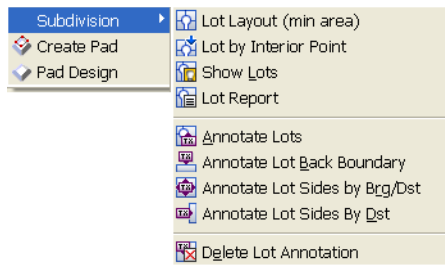


Figure 13-87. Design Menu – Subdivision Options

In This Section:

- “Lots Layout”
- “Lot By Interior Point” on page 13-81
- “Show Lots” on page 13-84
- “Lot Report” on page 13-85
- “Annotation Lots” on page 13-86
- “Annotate Lot Back Boundary” on page 13-87
- “Annotate Lot Sides by Brg/Dist” on page 13-88
- “Annotate Lot Sides By Distance” on page 13-89
- “Delete Lot Annotation” on page 13-91
- “Pad Design” on page 13-94

Lots Layout

Use the Lot Layout command to create lots within a specified area. It computes the position of the last boundary line to complete a lot of a known area. This requires an initial closed area, which is divided into two or more lots of a specified area.

This command uses two major methods to create the boundary lines, defining the closing boundary of the lot using either a line of specified bearing or a line pivoted about an existing point.

To create lots within a specified area, do the following:

1. Click **Design** ▶ **Subdivision** ▶ **Lot Layout (min area)**.

- Click in an enclosed area. A confirmation message appears (Figure 13-88).

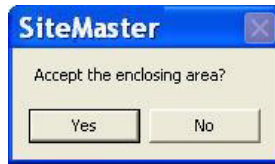


Figure 13-88. Confirmation Message

- Click **Yes**.
- Select the starting edge. This will be part of the first lot and approximately opposite to where the new boundary line will be created.
- Enter the lot area in the **Area** field of the input bar (Figure 13-89), and press **Enter**.



Figure 13-89. Lots - Input Bar

See also “Pivot” and “Frontage” on page 13-80.

- Enter another lot area in the **Area** field, and press **Enter**. Repeat this step as necessary.
- When you have finished creating lots, select **Create** from the right-click shortcut menu.

Pivot To pivot the lot boundary around an existing point on the initial closed area:

- Select the **Pivot** option of the input bar (Figure 13-90).



Figure 13-90. Lot Input Bar - Pivot

- Click on an arc or a point (Figure 13-91 on page 13-80).

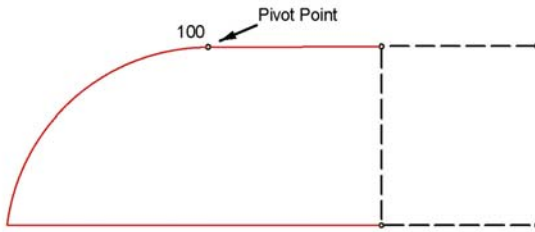


Figure 13-91. Pivot Point 100 for Computation of Second Lot

In Figure 13-91, the second lot is set to pivot around the point 100 at the node where the arc joins the top boundary line.

3. Enter the lot area, and then press **Enter**.
4. When the calculations are complete, right-click and select **Save Lots** from the shortcut menu.

A confirmation message appears (Figure 13-92).



Figure 13-92. Confirmation Message

5. Click **Yes** to save the remaining area as lots or click **No** to save only the calculated lots.

Frontage To create a frontage boundary line:

1. Check mark the **Frontage** box on the input bar (Figure 13-93), and enter a value in the field next to it. This value represents the boundary width at the front of the lot. (This could also work for a back boundary.)



Figure 13-93. Lot Input Bar – Frontage

2. Click on the boundary line representing the frontage (Figure 13-94 on page 13-81), and then press **Enter** or select **Create** from the right-click shortcut menu.

A new lot is formed with the nominated frontage, but with a modified area using either the Bearing or Pivot setting.

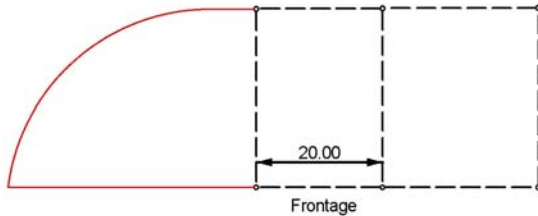


Figure 13-94. Frontage Set and Bearing Defined

If the pivot option is used with a frontage, the pivot point does not have to exist. The minimum area is used with the frontage width to compute the closing line of the lot around a pivot point created by the nominated frontage distance.



The right-click shortcut menu provides several commands for creating lots.

Lot By Interior Point

Use the Lot by Interior Point command to create lots from closed areas or polygons and save them in the MAGNET Office dataset. Each lot will have a unique name or number using a similar convention to that used for numbering points. A lot is a polygon entity with additional information entered to identify the lot. A lot may contain arcs. This command does not apply to polylines.

Do the following to use this command:

1. Click **Design** ► **Subdivision** ► **Lot by Interior Point**.
2. Click in an enclosed area. The full extents of the lot must be displayed on the screen for the area to be selected.

A confirmation message may appear (Figure 13-95 on page 13-82).



Figure 13-95. Confirmation Message

3. Click **Yes**. See also “Convert Polygon to a Lot” on page 13-84. The *Edit Lot* dialog box appears (Figure 13-96).

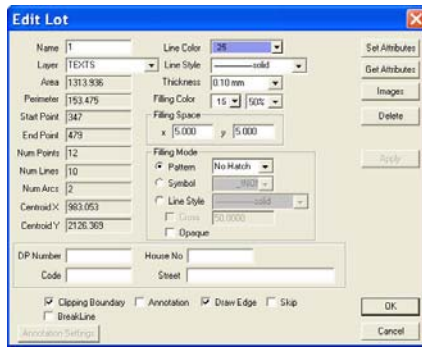


Figure 13-96. Edit Lot Dialog Box Used to Allocate a Lot Number and Other Parameters

4. Modify the properties as required, and click **OK** to save the lot. Future lots will now be inserted in a similar way until all of the lots have been saved. Table 13-12 describes the edit lot fields.

Table 13-12. Edit Lot Field Descriptions

Field Name	Field Descriptions
Name	Displays the next available lot name in the sequence. Modify as necessary, but only use numbers or letters.
Layer	The Layer is taken from the setting on the Property Bar.
Area	Displays the area enclosed within the lot.
Perimeter	Displays the perimeter around the lot boundaries.
Start Point/End Point Num Points/Num Lines Num Arcs	Definition of the lot
Centroid X & Y	Coordinate value of the centroid of the lot

Table 13-12. Edit Lot Field Descriptions (Continued)




Field Name	Field Descriptions
Line Color	The color of the lines defining the edge of the lot. Initially taken from the current settings on the Property Bar but can be modified as required.
Line Style	The line style of the lines defining the edge of the lot. Initially taken from the current settings on the Property Bar but can be modified as required.
Thickness	The thickness of the lines defining the edge of the lot. Initially taken from the current settings on the Property Bar but can be modified as required.
Filling Color	The color of the fill for shading and patterning, and the shading percentage of the fill color.
Filling Space	The X and Y spacing for use with the Symbol fill or Line Style hatching.
Pattern	Select a pattern from the drop-down menu. The line color and fill color must be different for the patterns to display. Choose a single color from the drop-down menu for a shaded lot.
Symbol	Select a symbol from the drop-down menu. The point symbols entity must be selected in the General Settings (Hot Button ) for the symbols to display.
Line Style	Select a line style from the drop-down menu. The line style entity must be selected in the General Settings (Hot Button ) for the line style to display. The line angle is taken from the angle setting in the next field.
Cross	This enables cross hatching
DP Number	Enter a DP number if required
House No	Enter the House No for this lot if required
Code	Enter a code to categorize this lot if required
Street	Enter the street name if required
Clipping Boundary	Select this check box to use the lot as a contour clipping boundary. The contours will not exist within the lot.
Annotation	Select the check box to turn on the lot annotation. The lot details annotation must be selected in the General Settings (Hot Button ) dialog box for the annotation to display.
Draw Edge	Select this check box to draw the edge of the lot on the current layer.

Table 13-12. Edit Lot Field Descriptions (Continued)

Field Name	Field Descriptions
Skip	Select this check box to apply the settings made for the first lot to additional lots in this session.

Additional Information

- The lot shape can be updated by moving the nodes of the lot. If a common mode is moved, then only one lot at a time is updated. Drag the node for the second lot to snap on the moved node of the first lot.
- If the lot is deleted, then the nodes (points) are retained.
- If the lot is not highlighted when you click inside the closed area, make sure all of the boundary lines are segmented at the corner points of the lot.

Convert Polygon to a Lot If the definition of the lot boundary is difficult to insert a lot, then a message appears, asking if you want to convert the polygon to a lot (Figure 13-97).



Figure 13-97. Convert Polygon to a Lot

Click **Yes**, and the *Edit Lot* dialog box appears (see Figure 13-96 on page 13-82).

Show Lots

Use the Show Lots command to highlight all of the currently defined lots in the project and fill them with a temporary hatching pattern (Figure 13-98 on page 13-85).

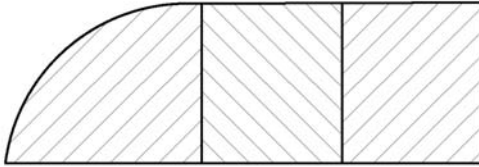


Figure 13-98. Lot Hatching

To use this command, click **Design ▶ Subdivision ▶ Show Lots**. Any lot on the active layer is filled with a hatch pattern on the screen. This hatching disappears if the screen is refreshed using the **redraw**, **regen**, **pan**, or **zoom** commands.

Lot Report

Use the Lot Report command to list all of the lots on the active layers or the lots in the current selection. The report contains full details of the lots, including the bearing and distances of the line boundaries and the arc details of the arc boundaries.

Do the following to use this command:

1. If required, define a selection of lots.
2. Click **Design ▶ Subdivision ▶ Lot Report**.
The *List Lots* dialog box appears (Figure 13-99).



Figure 13-99. List Lots Dialog Box

3. Complete the **List Lots** fields. For field descriptions, see Table 13-13 on page 13-86.
4. Click **OK** to produce the report.

A report displays (Figure 13-100). Each lot is listed with its details. The report may be read on the screen, printed, or saved to a report (.rep) file as a record of the defined lot(s).

SiteMaster - List Lots Jul 02, 2010 11:06AM Page: 1

Job Name: Parkdemo

Lot: 1		DP:	Street Name:		Area: 0.010 (acres)		Code:	
House Number:	Line	Bearing	Distance	Arc Length	Radius	PENO	Easting	Northing
	1	142°12'58"	20.354			1	3229.338	3215.532
	2	61°26'55"	15.138			2	3247.188	3192.228
	3	322°32'58"	26.898			48	3240.485	3199.464
	4	243°09'10"	15.216			48	3242.914	3222.404

Lot: 2		DP:	Street Name:		Area: 0.010 (acres)		Code:	
House Number:	Line	Bearing	Distance	Arc Length	Radius	PENO	Easting	Northing
	1	142°12'58"	20.896			48	3242.914	3222.404
	2	61°26'55"	15.138			47	3240.485	3199.464
	3	322°32'58"	26.431			49	3271.997	3206.816
	4	243°09'10"	15.461			50	3256.709	3229.386

Lot: 3		DP:	Street Name:		Area: 0.007 (acres)		Code:	
House Number:	Line	Bearing	Distance	Arc Length	Radius	PENO	Easting	Northing
	1	142°12'58"	20.431			50	3256.709	3229.386
	2	61°26'55"	7.863			49	3271.997	3206.816
	3	318°23'15"	27.739			3	3280.906	3210.574
	4	243°09'10"	14.336			4	3269.500	3229.386

Figure 13-100. Lot Report

Table 13-13. List Lots Field Descriptions

Field Name	Field Description
All Lots	This option is chosen if no lots are selected prior to using this command.
Current Selection	This option is chosen if lots are included in the current selection.
Code	Enter a code or a range of codes to filter the list with only the lots labelled with this code or code range. A range can be defined with hyphens; for example, 12-16 or abc-hij. Use commas to specify individual items; for example, 16,18,20.
DP Number	Enter a DP Number or a range of DP numbers to list only the lots labelled with this DP number or number range. A range can be defined with hyphens; for example, 12-16 or abc-hij. Use commas to specify individual items; for example, 16,18,20.
House Number	Enter a house number or a range of house numbers to list only the lots labelled with this number or number range. A range can be defined with hyphens; for example, 12-16 or abc-hij. Use commas to specify individual items; for example, 16,18,20.

Annotation Lots

Use the Annotation Lots command to annotate the selected lots using the parameters set up in the current *Lot Annotation Settings* dialog

box (see “Survey View: Settings Menu” on page 8-1 for details). An example of lot annotation is shown in Figure 13-101 below.

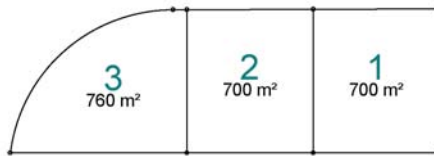



Figure 13-101. Lots Annotated with a Lot Number and Area

To use the Annotation Lots command, do the following:

1. Make sure the **Lot Details** option is selected on the *Display* tab in the **General Settings (Hot Button)**  dialog box.
2. Define a selection of lots. (You can click and drag a selection window over the lots.)
3. Click **Settings** ▶ **Annotation Settings** ▶ **Lot Annotation Settings**.
4. On the *Lot Annotation Settings* tab, modify the properties as needed, and click **OK**.

Annotate Lot Back Boundary

Use the Annotate Lot Back Boundary command to annotate the collinear back boundary of the selected lots with either one bearing value or one bearing value and the total distance (Figure 13-102).

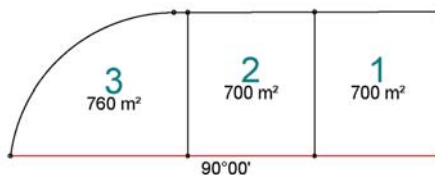



Figure 13-102. Lot Back Boundary Selected for the Three Lots

To use the Annotate Lot Back Boundary command, do the following:

1. Define a selection of lots. (You can click and drag a selection window over the lots.)

2. Make sure the **Back Boundary** option is selected in the **Lot Annotation Settings** dialog box (**Settings ▶ Annotation Settings**).
3. Click **Design ▶ Subdivision ▶ Annotate Lot Back Boundary**.
4. Select the boundary of one lot in the selection, starting at one end of the boundary line. The boundary line is highlighted in red.
5. Select an adjacent collinear boundary and continue selecting all of the lot boundaries that comprise the back boundary.
6. When the selection is complete, right-click and select **Annotate** from the shortcut menu.

The selected back boundary is annotated with either the bearing value positioned at the center of the selected lot boundaries or the bearing and distance value spread along the boundary.

MAGNET Office displays the annotation if the **Lot Details** option is selected on the **Display** tab in the **General Settings** (Hot Button ) dialog box.



The annotation format depends on the current settings in the **Lot Annotation Settings** dialog box (**Settings ▶ Annotation Settings**).

Annotate Lot Sides by Brg/Dist

Use the Annotate Lot Sides by Brg/Dist command to annotate nominated boundaries of selected lots (Figure 13-103) with a bearing and distance using the parameters set up in the **Settings ▶ Annotation Styles** dialog box (**Lot Annotation** tab).

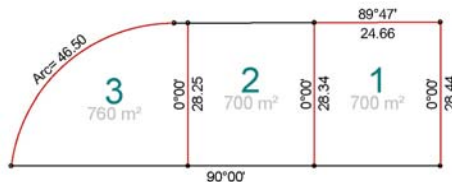



Figure 13-103. Lots with Boundaries Annotated with Bearing and Distance

To use the Annotate Lot Sides by Brg/Dist command, do the following:

1. Define a selection of lots. (You can click and drag a selection window over the lots.)
2. Click **Design ▶ Subdivision ▶ Annotate Lot Sides by BrgDist**.
3. Select a lot boundary.
The boundary is highlighted in red.
4. Continue to select all of the boundary edges to be annotated.
5. When the selection is complete, right-click and select **Annotate**.
The selected boundary edges are annotated with bearing and distance values positioned at the center of the selected boundaries.

MAGNET Office displays the annotation if the **Lot Details** option is selected on the *Display* tab in the *General Settings (Hot Button *) dialog box.

Additional Information

- The annotation format depends on the current settings in the **Settings ▶ Annotation Styles** dialog box.
- Make sure the **Lines** setting in the *Annotation Settings* dialog box (*Line Annotation Setting* tab) is selected so you can annotate the lot edges by bearing and distance.
- Arc boundaries may also be selected and annotated with the arc length if the **Arcs** setting is selected in the **Settings ▶ Annotation Styles** dialog box (*Arc Annotation Setting* tab).

Annotate Lot Sides By Distance

Use the **Annotate Lot Sides by Dst** command to annotate nominated boundaries (Figure 13-104) of selected lots with a distance using the

parameters set up in **Settings ▶ Annotation Styles (Distance Annotation Settings** tab).

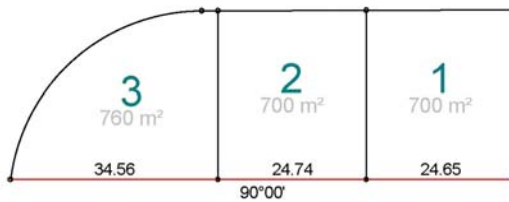



Figure 13-104. Lots with Back Boundaries Annotated with Distance

To use the Annotate Lot Sides by Dst command, do the following:

1. Define a selection of lots. (You can click and drag a selection window over the lots.)
2. Click **Design ▶ Subdivision ▶ Annotate Lot Sides By Dst**.
3. Select a lot boundary.
The boundary is highlighted in red.
4. Continue to select all of the boundary edges to be annotated.
5. When the selection is complete, right-click and select **Annotate**.

The selected boundary edges are annotated with distance values only positioned at the center of the selected boundaries.

MAGNET Office displays the annotation if the **Lot Details** option is selected on the **Display** tab in the **General Settings** (Hot Button ) dialog box.

Additional Information

- The annotation format depends on the current settings in the **Lot Annotation Settings** dialog box (**Settings ▶ Annotation Styles**). The **Annotation Settings** dialog box appears.
- Make sure the **Lines** setting in the **Annotation Settings** dialog box is selected to enable annotation of the lot edges by bearing and distance.
- Arc boundaries may also be selected and annotated with the arc length if the **Arcs** setting is selected in the **Annotation Settings** dialog box.

Delete Lot Annotation

To remove specified lot annotations from a selection of lots, do the following:

1. Define a selection of lots. (You can click and drag a selection window over the lots.)
2. Click **Design ▶ Subdivision ▶ Delete Lot Annotation**.
3. Select the edge or vertex of a lot.
4. Right-click and select the appropriate deletion command from the shortcut menu (Figure 13-105 on page 13-91).

The lot annotation will be removed.

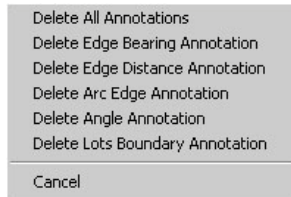


Figure 13-105. Popup Menu

5. Continue to select edges and remove the annotation from lots as required. (You can select several lot sides at once.)

Create Pad

Use the Create Pad command to create a building pad (Figure 13-106). The pad can be created from existing points, lines, strings, or polygon, or from new points created in this routine.

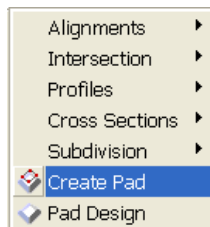


Figure 13-106. Design Menu – Create Pad Options

The pad is saved with a unique name and is used in the **Design ▶ Pad Design** command. You can create the pad with no elevation or with elevations that are relative to each other to define a surface. A complex pad may also include points with elevations in the body of the polygon, so that a surface model of the pad can be created.

To create a building pad, do the following:

1. Click **Design ▶ Create Pad**.

The input bar and prompt line appear (Figure 13-107).



Prompt Line

Figure 13-107. Create Pad Input Bar and Prompt Line

2. Select either:

- A start point and additional points and lines to define the pad.
- An existing line to follow a path around the pad shape, OR
- One leg of an existing polygon.

3. Click the next point or line with the mouse or enter coordinates in the input bar.

4. Continue locating points until all of the points for the pad are chosen. When the start point is selected again, the pad closes and the **Edit Pad** dialog box appears (Figure 13-108).

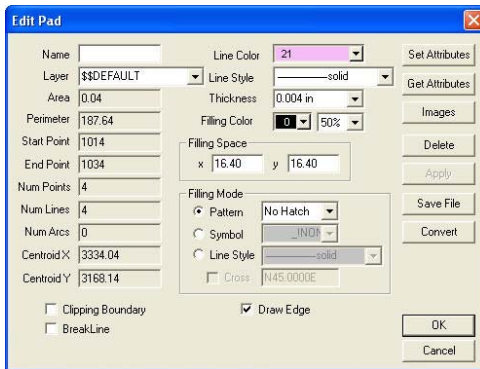


Figure 13-108. Edit Pad Dialog Box

5. Modify the properties as needed, and then click **OK** to create the pad. For a description of fields, see Table 13-14 on page 13-93.
6. To modify the pad, move the nodes while the pad is selected.



If the pad is deleted, then the nodes (points) and lines are retained.

Table 13-14. Edit Pad Field Description



Field Name	Field Description
Name	Enter a name for the pad.
Layer	Defines the layer on which the pad is created.
Area	Displays the area enclosed within the pad.
Perimeter	Displays the perimeter around the pad.
Start/End Points Num Points/Lines Num Arcs	Definition of the pad.
Centroid X & Y	Coordinate value of the centroid of the pad.
Line Color	Defines the line color of the pad edge.
Line Style	Defines the line style of the pad edge.
Thickness	Defines the line thickness of the pad edge.
Filling Color	The color of the fill for shading and patterning, and the shading percentage of the fill color.
Filling Space	The X and Y spacing for use with the fill or patterning.
Pattern	Choose Pattern to activate the drop-down menu, and then select a pattern or select No Hatch . The line color and fill color must be different for the patterns to display.
Symbol	Choose Symbol to activate the drop-down menu, and then select a symbol. The Point Symbols check box must be selected in the General Settings (Hot Button ) for the line style to display. The line angle is taken from the angle setting in the next field.

Table 13-14. Edit Pad Field Description (Continued)

Field Name	Field Description
Line Style	Choose Line Style to activate the drop-down menu, and then select a line style. The line style entity must be selected in the General Settings (Hot Button ) for the line style to display. The line angle is taken from the angle setting in the next field.
Clipping Boundary	Used as a contour clipping boundary. The contours will not exist within the boundary.
Breakline	Creates a breakline form a boundary.
Draw Edge	Draws the edge of the pad on the current layer.

Pad Design

Use the Pad Design command to compute the elevation of a building pad and slope positions at a defined grade from the pad to a nominated existing surface. A new surface is created for the pad and slopes. This surface may be merged with the existing surface to create an overall finished surface. A volumes computation for cut and fill volumes is displayed during the design process with the option to position the pad at a level where the cut volume equals the fill volume (Figure 13-109).



Figure 13-109. Design Menu – Pad Design

The pad may have its own surface model to represent the grade changes across the pad.

A pad may be saved in a project ready for use or a previously saved pad file may be imported into the current project.

Before using the Pad Design command, make sure all the components required for the computation have been suitably defined (Figure 13-110).

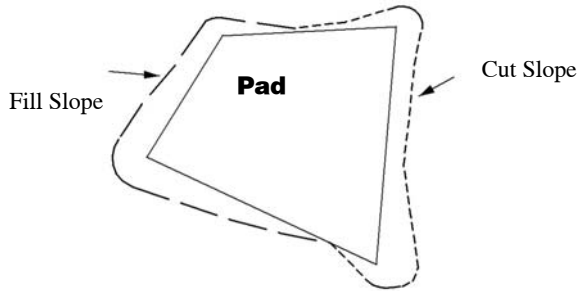


Figure 13-110. Finished Pad with Levels Set to Provide a Balance of Cut and Fill Volumes

To use the Pad Design command, do the following:

1. Click **Design ► Pad Design**.

The *Pad Design* dialog box appears (Figure 13-111).

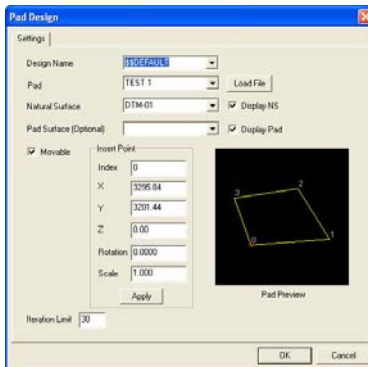


Figure 13-111. Pad Design Dialog Box

The only tab available is the *Settings* tab, unless there is already a pad saved in the project.

2. If a pad surface exists, select it from the *Pad* drop-down menu. If there is no pad available in the current project, then click **Load** to import a previously saved pad. However, if a pad exists in the project, it will be selected and displayed as shown on the *Settings* tab. If you have already

created a pad in a previous session, then the pad will display and there will also be a second tab called **Design Parameters**.

3. Modify the other properties as necessary, and then click **OK**. For a field description, see Table 13-15 below.

Table 13-15. Pad Design Field Descriptions

Field Name	Field Description
Design Name	The first pad design in a project is set to \$\$DEFAULT. You can change this name.
Pad	If a pad exists in the current project, then it will be listed. If more than one pad exists in the current project, then select the required pad from the list. If no pad exists in the current project, click Load to import a pad from a file.
Load	Imports a saved pad file.
Natural Surface	Select the existing surface to be used for the slope intercepts.
Display NS	Displays the triangular mesh in the design process window.
Pad Surface (Optional)	Select an existing surface that represents the grades across the pad (optional). If no surface exists, then a plane surface is computed from the relative levels of the points defining the pad edges.
Display Pad	Displays the triangular mesh of the pad in the design process window.
Movable	Enables movement of the pad to another position. This is useful when positioning a standard pad to different locations. If the pad is defined in its true coordinate position, then click to clear the check box.
Insert Point	This is one point on the pad edge.
Index	This defaults to 0 and is the point on one corner of the pad. It is highlighted in red in the Pad Preview . To change this number, click on a new point in the Pad Preview and click Apply in the Insert Point box.
X, Y, Z	Current settings for the coordinates and level of the insert point. If the pad has been loaded, the insert point can be defined for this project by entering suitable coordinates in these fields, and then clicking Apply .

Table 13-15. Pad Design Field Descriptions (Continued)

Field Name	Field Description
Rotation	Rotates the pad around the insert point. Enter a rotation angle.
Scale	Changes the pad scale. Enter a value in this field. The pad scale is modified based on the insert point position.
Apply	Applies changes made to the insert point settings
Iteration Limit	This defaults to 30. It may need to be increased for complex pads, where more calculations are needed to position the pad and compute the slop intercepts and volumes. This is important when computing equal cut/fill volumes.

Movable To move the pad, do the following:

1. Check mark the **Movable** box.
The Design View is displayed with the pad outline attached to the pointer.
2. Position the pad with the pointer or enter the coordinates in the input bar.
3. If you use the pointer to position the pad, then left-click to place the pad.
4. Click **OK** on the input bar.
The *Pad Design* dialog box reappears with the **Design Parameters** tab open (Figure 13-112).

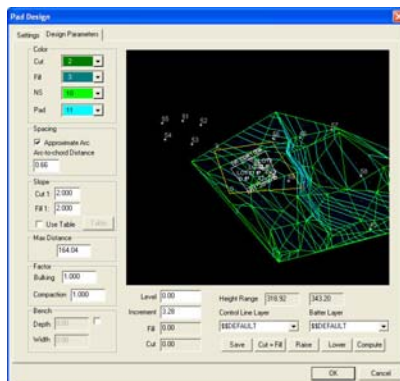


Figure 13-112. Pad Design - Design Parameters Tab

- Modify the properties as necessary, and click **Compute**. See “Design Parameters” on page 13-99 and “Cut=Fill” on page 13-99. For field descriptions, see Table 13-16 below.

Table 13-16. Design Parameters Field Descriptions

Field Name	Field Description
Color	
Cut	The cut is displayed in the color selected from this drop-down menu.
Fill	The fill is displayed in the color selected from this drop-down menu.
NS	The natural surface is displayed in the color selected from this drop-down menu.
Pad	The pad is displayed in the color selected from this drop-down menu.
Slope	
Cut1	Use the default or enter the horizontal component of the slope in a cut situation.
Fill1	Use the default or enter the horizontal component of the slope in a fill situation
Max Distance	Use the default or enter the maximum distance in meters from the pad perimeters that the routine will search for a slope intercept within the existing surface.
Factor	
Bulking	Enter a bulking factor to apply to the calculation of volumes in cut. Enter a value greater than 1.00 to increase the cut volume by this ratio. Leave at 1.000 for no compaction of volumes.
Compaction	Enter a compaction factor to apply to the calculation of volumes in fill. Enter a value less than 1.00 to decrease the fill volume by this ratio. Leave at 1.000 for no compaction of volumes.
Bench	
Depth	Enter a depth to nominate how often a bench will be calculated in the side slope.
Width	Enter a value to define the width of the bench.

Table 13-16. Design Parameters Field Descriptions (Continued)

Field Name	Field Description
Level	Start with an approximate level for the insert point on the pad. Click Compute to check the volumes at this level and display the slope lines and cut/fill sections.
Increment	Enter the increment in meters that the pad will be raised or lowered by clicking Raise or Lower .
Fill	Displays the computed fill volume.
Cut	Displays the computed cut volume.
Control Line Layer	Select a layer from the drop-down menu. The data on this layer is displayed in the preview window as a guide to the position on the pad and the extents of the computed slopes. Data on a specific layer may be critical to these computations. Any layer that is in the project may be displayed.
Batter Layer	Select a layer from the drop-down menu or enter a new layer name. This layer holds the cut and fill batter lines and points when the computation is complete, and the data is saved in the Survey View.

Design Parameters The various fields on this tab allow you to set a range of properties to compute and to test a suitable final layout for the pad. MAGNET Office calculates the volumes for the defined side slopes, the height of the pad, and its surface if used, relative to the levels on the natural surface.

After you have defined the properties, click **Compute** to calculate initial cut and fill volumes. Use the buttons to modify the pad levels or calculate a position where the cut and fill volumes are equal.

Be aware that the slopes may only be computed as long as the side slopes and levels chosen allow for the slope to be inside the extents of the available natural surface. Error messages will appear if a computation cannot be made.

The maximum distance defined may not relate to the slopes and pad level. You may need to modify the angular setting, especially on complex shaped pads. You may need to adjust the spacing between the slope intercept (cross section) calculations.


Cut=Fill The Cut=Fill computation is an interactive procedure and may take some time to achieve a result if complex surfaces are

involved. The iteration limit on the **Settings** tab may need to be increased. Once the pad position and volumes suit the project design, the pad position levels, surface, and slopes should be saved to the Survey View by clicking **Save**.

When the data has been saved to the Survey View, click **OK** on the *Pad Design* dialog box (see Figure 13-111 on page 13-95) to close it and return to the Design View.

Road View/Edit

When cross sections have been extracted, a Road is listed by its alignment name.

Click **Design ▶ Road View/Edit** or click the **Roads** module icon .

The Road is selected and opened to display the Roads View, which is divided into three panes of data.

- Alignment pane – displays the horizontal alignment and design strings. The plan data from the Survey View may also be shown.
- Cross Section pane – displays the cross section for one chainage. Several surfaces may be shown.
- Profile pane – displays the long section for the natural surface and the design profile with vertical intersection points for the reference alignment or any one selected string.



The display in each window is dependent on settings selected in the **General Settings** dialog box (press the **Hot Button** located at the bottom of the screen).

See “Roads View Editor” on page 18-1 for details on this menu option.

Survey View: Surface Menu

This section describes the different Survey View menu commands on the Surface menu located on the main menu bar (Figure 14-1).



Figure 14-1. MAGNET Office – Surface Menu

In This Section:

- “DTM Settings” on page 14-2
- “Contour Settings” on page 14-4
- “Create DTM (Digital Terrain Model)” on page 14-5
- “Create DTM by Depth Range” on page 14-14
- “Compute Elevations” on page 14-15
- “Merge DTM” on page 14-29
- “Update DTM” on page 14-30
- “Trim DTM” on page 14-31
- “DTM Tools” on page 14-31

- “Create Boundary” on page 14-63
- “Create Enclosing Boundary” on page 14-67
- “Create Boundary From Polygons” on page 14-70
- “Create Boundary List” on page 14-71
- “Edit Boundary Lists” on page 14-73
- “Compute Volume” on page 14-74
- “Batch Volumes” on page 14-80
- “Quick Volume” on page 14-82

DTM Settings

Use the DTM Settings command to control the display of the Digital Terrain Models (surface models) in the current project. DTMs can be edited and deleted and the mesh and contours can be turned on or off.

To control the display of DTM in the current project, do the following:

1. Click **Surface ► DTM Settings**. The *DTM* dialog box appears (Figure 14-2).

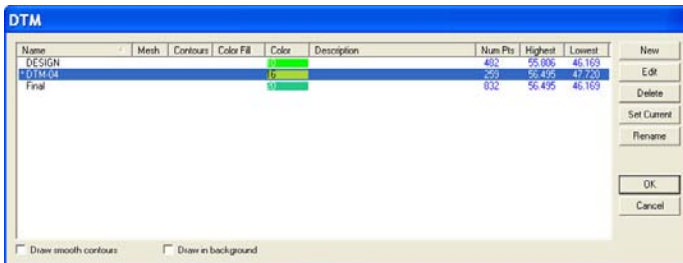


Figure 14-2. DTM Window

The dialog box lists all of the DTMs in the current project. The DTM marked with an asterisk is the current DTM.

2. To change the current DTM, select one from the list and press **Set Current**.

3. Modify the properties as necessary, and press **OK**. For a description of the fields, see Table 14-1 below.

Table 14-1. DTM Settings Field Descriptions

Field	Description
Name	Name of the DTM. The name was defined when the DTM was created.
Mesh	Select this field to display the triangular mesh in the Survey View. Click to clear the check mark.
Contours	Select this field to display the contours in the Survey View. Click to clear the check mark.
Color Fill	Double-click with the mouse to display the fill color selected on the display pane of the DTM Settings dialog box (Figure 14-3 on page 14-4).
Color	Double-click with the mouse to display the color selected on the display pane of the DTM Settings dialog box (Figure 14-3 on page 14-4).
Description	Displays the description, if any, used for the DTM.
Num Pts	Number of points in this DTM.
Highest	Highest elevation in this DTM.
Lowest	Lowest elevation in this DTM.
New	Press to display the DTM Settings dialog box (Figure 14-3 on page 14-4)/
Edit	Edits the highlighted DTM in the DTM dialog box (Figure 14-2 on page 14-2).
Delete	Deletes the highlighted DTM.
Set Current	Press to change the current DTM to another one.
Rename	Renames the highlighted DTM.
OK	Press to save.
Cancel	Press to quit the routine.

New DTM To create a new DTM, click **New** in the DTM window. For details, see “Create DTM (Digital Terrain Model)” on page 14-5.

Edit DTM To edit a DTM, select the DTM you want to edit in the *DTM* dialog box, and click **Edit**.

The *DTM Settings* dialog box appears (Figure 14-3 on page 14-4).



Figure 14-3. DTM Settings Dialog Box

Modify the properties as necessary, and then click **OK**. For field descriptions, see “General Tab” on page 14-9, “Contour Tab” on page 14-10, and “Contour Label Tab” on page 14-11.

When contours are created from a mesh, the contour interval is interpolated along each triangle side. These points are joined with straight lines causing the contours to appear jagged (Figure 14-4).

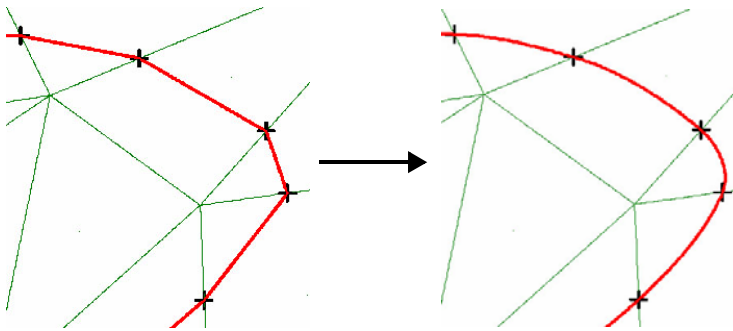


Figure 14-4. Smoothed Contours

Contour Settings

Click **Surface ▶ Contour Settings**. The *Contour Settings* dialog box appears (Figure 14-5 on page 14-5).

Do the following to create, modify, rename, delete, or copy a contour setting.

- **New** – press to create a new contour file.

- **Modify** – press to modify the highlighted file.
- **Rename** – press to rename the highlighted file.
- **Delete** – press to delete the highlighted file.
- **CopyFromLib** – press to copy a contour setting from the Library.

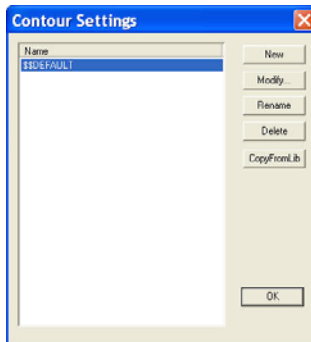



Figure 14-5. Contour Settings Dialog Box

Create DTM (Digital Terrain Model)

Use the Create DTM command to form a Surface Model or DTM as a triangular mesh from the points in the database that have elevations and are flagged as contourable. The extents of the mesh can be based on a selection of points or all points in the project on the active layers. The command requires a minimum of five contourable points to form a triangular mesh.

Make sure the points for the surface definition are displayed on the active layers and these points have their **Contourable** options selected on the **Default Attributes** tab in the **General Settings** (Hot Button ) dialog box.

To use the Create DTM command, do the following:

1. Click **Surface ▶ Create DTM**.

The *Create DTM - Selection* dialog box appears (Figure 14-6 on page 14-6).

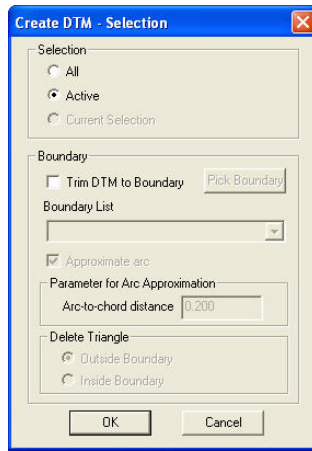
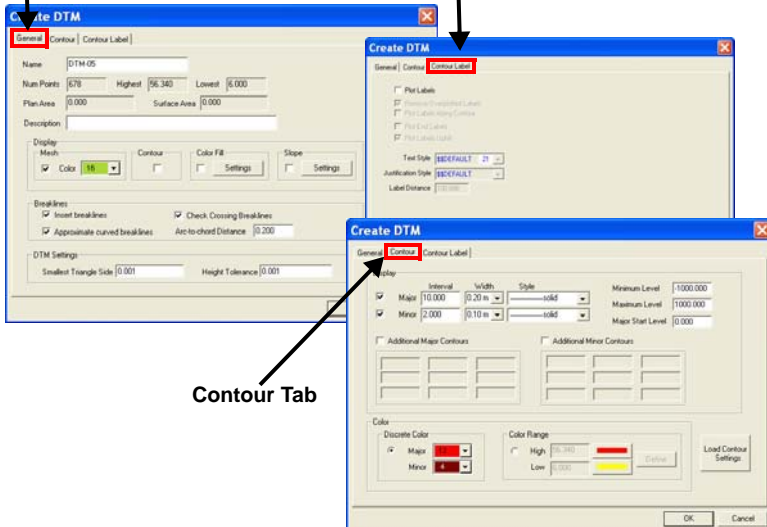


Figure 14-6. Create DTM – Selection Dialog Box

2. Define the properties as needed, and then click **OK**. The *Create DTM* dialog box appears with the *General* tab open (Figure 14-7).

General Tab

Contour Label Tab



Contour Tab

Figure 14-7. Create DTM Dialog Box

- Click each tab and modify the properties as needed. For field descriptions, see “General Tab” on page 14-9, “Contour Tab” on page 14-10, and “Contour Label Tab” on page 14-11.
- Click **OK**.

The error report confirmation message appears (Figure 14-8).



Figure 14-8. Error Report Confirmation Message

- Click **Yes** to display the report (Figure 14-9).

```

File Edit View Window Help
SiteMaster - Jul 12, 2010 12:12PM Page: 1

Create DTM Report

Check Breaklines

String (86, 245) ignored - a point of the breakline has no height.
String (796, 807) ignored - it intersects with other breakline.
String (636, 667) ignored - it intersects with other breakline.
Line (15, 163) ignored - it intersects with other breakline.
Line (24, 25) ignored - it intersects with other breakline.
String (768, 795) ignored - it intersects with other breakline.
String (856, 865) ignored - it intersects with other breakline.
String (828, 859) ignored - it intersects with other breakline.
String (924, 955) ignored - it intersects with other breakline.
String (892, 923) ignored - it intersects with other breakline.
Line (20, 21) ignored - it intersects with other breakline.
String (700, 733) ignored - it intersects with other breakline.
String (732, 762) ignored - it intersects with other breakline.
String (603, 635) ignored - it intersects with other breakline.
String (668, 699) ignored - it intersects with other breakline.

Breakline Insertion Report

Summary

678 points used in the triangulation.
Highest point elevation : 56.340
Lowest point elevation : 6.000
Total number of triangles in the DTM: 1574
Total plan area of the DTM : 37028.937
Total surface area of the DTM : 42172.715


```

Figure 14-9. Error Report

The report shows the crossing breaklines, duplicated points, points with no levels at the end of the breaklines, and any other problems. The relevant points and lines are listed in the report (Figure 14-10 and Figure 14-11 on page 14-8).

6. Check the report to ensure the DTM is accurate. Click **Surface ▶ DTM Tools ▶ Check Breakline** to work through any crossing breaklines (Figure 14-12 on page 14-9).



Once the DTM is created, you can edit any of the properties set during creation. Click **Surface ▶ DTM Settings** or click the **DTM**  icon on the Status Bar and select a DTM from the list to edit.

The triangular mesh and contours are displayed in the Survey View.

The **Program Settings** are used to define duplicate points. The values set for the smallest triangle and the height tolerance are used for this procedure.

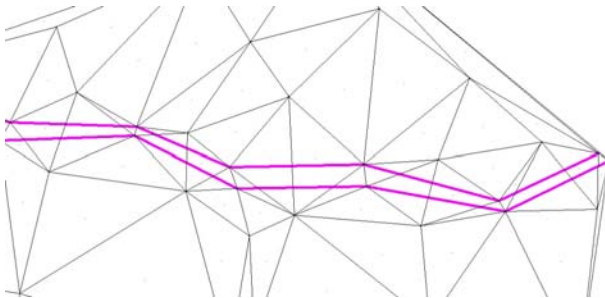


Figure 14-10. DTM Created as a Triangular Irregular Network With No Breaklines

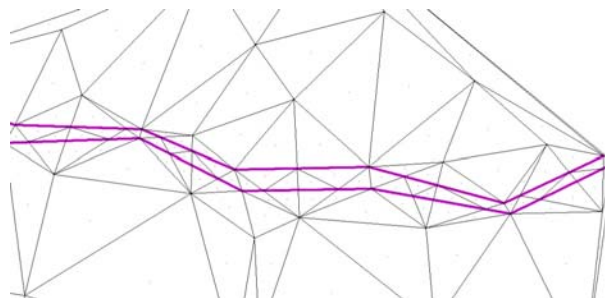


Figure 14-11. Breaklines Highlighted with Additional Triangles Added to Correct Surface

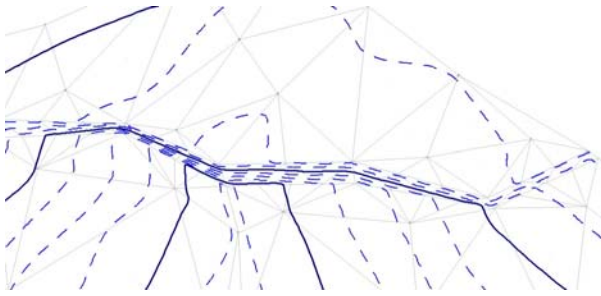


Figure 14-12. Display of Major and Minor Contours

General Tab For field descriptions, see Table 14-2.

Table 14-2. Create DTM – General Tab Field Descriptions

Field	Descriptions
Name	By default, the first DTM in a project is set as DTM-01. You can rename it. Additional DTMs will be named DTM-02, DTM-03, and so on.
Num Points	Displays the number of contourable points in the DTM.
Highest	Displays the highest elevation in the DTM.
Lowest	Displays the lowest elevation in the DTM.
Plan Area	Displays the horizontal area (L x W).
Surface Area	Displays the surface area, including slopes, not just the area dimensions.
Description	Enter an optional DTM description.
Display	
Mesh	Displays the created triangular mesh in the Survey View.
Color	Opens the color pallet (click on the drop-down entry box) and applies the chosen color to the triangular mesh.
Contour	Check mark this box to display the contours in the Survey View.
Color Fill	Press the Settings button to define the color of the fill for shading and patterning.

Table 14-2. Create DTM – General Tab Field Descriptions

Field	Descriptions
Slope	Defines the slope ranges and colors (press the Settings button). See also “Slope Settings” on page 14-12.
Breaklines	
Insert breaklines	Check mark this box to create additional triangles, so the triangle edges are formed along each breakline. If a breakline exists in the project, then this check box is automatically selected.
Approximate curved breaklines	Check mark this box to create additional triangles, so the triangle edges are formed along chords of the curved breaklines.
Check Crossing Breaklines	Check mark this box to insure the intersection is correct.
Arc-to-chord Distance	Enter the arc to chord distance for curved breaklines. The default is 0.200 m and is suitable for most models. Edit this value if necessary.
DTM Settings	
Smallest Triangle Side	Enter a parameter to check for duplicated points while creating a DTM.
Height Tolerance	Enter a parameter to compare points at the same position during the breakline process.

Contour Tab For field descriptions, see Table 14-3.

Table 14-3. Create DTM – Contour Field Descriptions

Field	Description
Display	
Major	Enter the major contour interval required. Set the line width and style, and select the check box to display the major contours.
Minor	Enter the minor contour interval required. Set the line width and style, and select the check box to display the minor contours.
Minimum Level/ Maximum Level	Enter the range of levels in which the contours are created. The range should be between the lowest and highest point elevation in the DTM.

Table 14-3. Create DTM – Contour Field Descriptions

Field	Description
Major Start Level	Enter the level where the first major contour is created.
Additional Major Contours/ Additional Minor Contours	In addition to the levels calculated by MAGNET Office, you can select either or both check boxes and specify the levels where the extra contours are created.
Color	
Discrete Color	Select this option to open the color palette and set a Major and Minor line color.
Color Range	Select this option, then press Define to change colors on the <i>Color Range Settings</i> dialog box (Figure 14-13 on page 14-12).
Low	Enter the lowest value of height range or accept the value for the lowest contourable point in the project. Press the color button to choose a color from the palette. This displays the lowest DTM elevation with the color you selected. See also “Color Range Settings” on page 14-12.
High	Enter the highest value of height range or accept the value for the highest contourable point in the project. Click the color button next to it to choose a color from the palette. This displays the highest DTM elevation with the color you selected. See also “Color Range Settings” on page 14-12.
Load Contour Settings	Press to copy the default settings from the project or saved settings from the library.

Contour Label Tab For field descriptions, see Table 14-4.

Table 14-4. Create DTM – Contour Label Field Descriptions

Field	Description
Plot Labels	Plots labels on the contour.
Remove Over plotted Labels	Removes overlapping contour labels.
Plot Labels Along Contour	Places the labels along the contour.
Plot End Labels	Plots labels at the end of the contour.

Table 14-4. Create DTM – Contour Label Field Descriptions (Continued)

Field	Description
Plot Labels Uphill	Places the labels.
Text Style	Defines the text style of the label.
Justification Style	Defines the justification style of the labels.
Label Distance	Defines the spacing between the labels along the contour.

Color Range Settings The DTM is created using the color range option with a gradual change in color for each minor contour interval from the highest to the lowest contour (Figure 14-13).

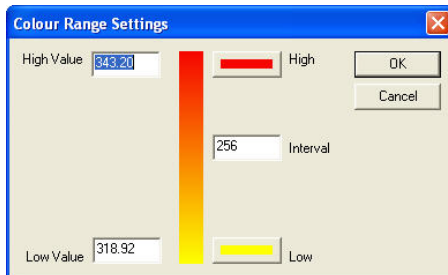


Figure 14-13. Color Range Settings

Set the minor contour interval to a very small value to display a full progression of the color range across the DTM (Figure 14-14).

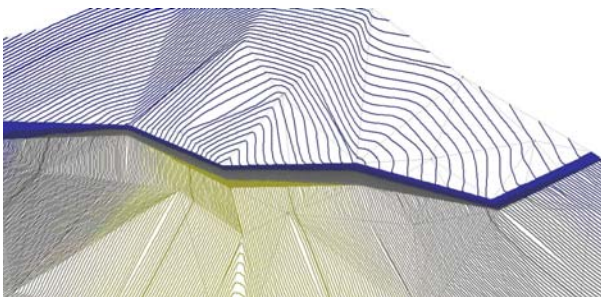


Figure 14-14. Color Range Shows Change in Levels

Slope Settings This option allows the slopes of the individual triangles to be analyzed (see Figure 14-7 on page 14-6, the “General Tab”). You can allocate different colors to the slope ranges with the

option to set the shape of the arrows that display the slopes (Figure 14-15 on page 14-13).

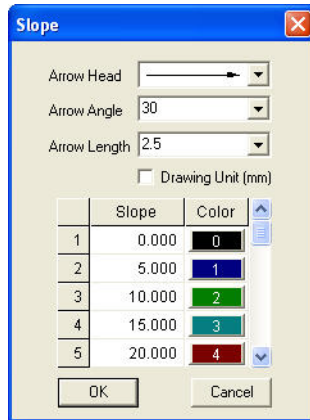


Figure 14-15. Slope Settings

1. Select the arrow head, angle, and length from the drop-down menus (Figure 14-16).
2. In the table, enter consecutive steps for the slope range with an appropriate color for each slope.
3. Click **OK** to apply the settings.

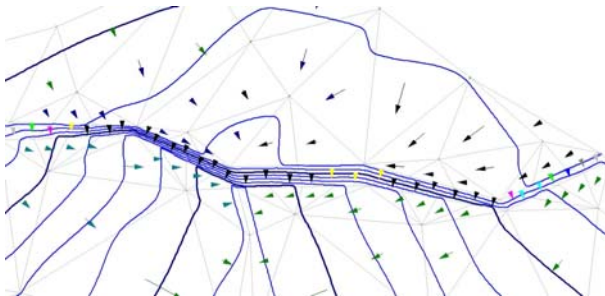


Figure 14-16. Slopes of Triangles Marked with Arrows

Create DTM by Depth Range

The Create DTM by Depth Range command creates a DTM from a point selected with a specified height difference from an existing DTM to create a new DTM. A reference DTM must be selected with minimum and maximum height above or below the specified reference DTM.

To create DTMs by depth range, click **Surface ► Create DTM by Depth Range**. The *Create DTM by Depth Range* dialog box appears (Figure 14-17).

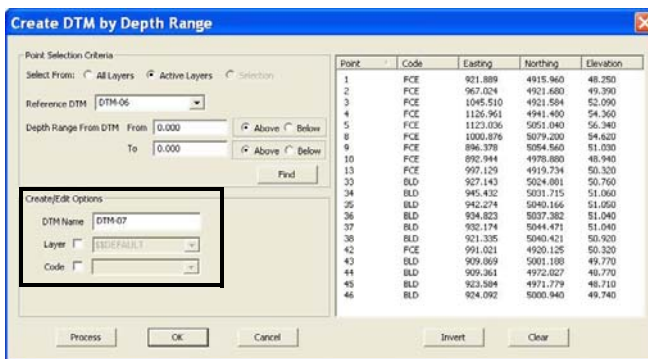


Figure 14-17. Create DTM by Depth Range

You can also move the selected points into a specified layer and re-code the points if necessary.

In the *Create/Edit Options* pane on the *Create DTM by Depth Range* dialog box (Figure 14-17), do the following to recode points:

- Place a check mark in the **Layer** box and select a layer name from the drop-down menu.
- Place a check mark in the **Code** box and select a code name from the drop-down list.

If these options are not check marked, the original points will remain and only the DTM will be created.

A DTM is then formed using the point selection by clicking the Process button. More points can be selected and additional DTMs can be processed without exiting.

Alternatively click **OK** to create only one DTM from selection.

The selection can be refined by highlighting only the required points listed on the right pane displaying the selected points using the **Invert** and **Clear** buttons.

- Press **OK** to save your work.
- Press **Cancel** to quit the routine.



This feature is an expansion of the “Select by DTM”, which allows you to only select points selected based on height range from a specified DTM.

Compute Elevations

Use the Compute Elevations commands to calculate elevations between entities (Figure 14-18).

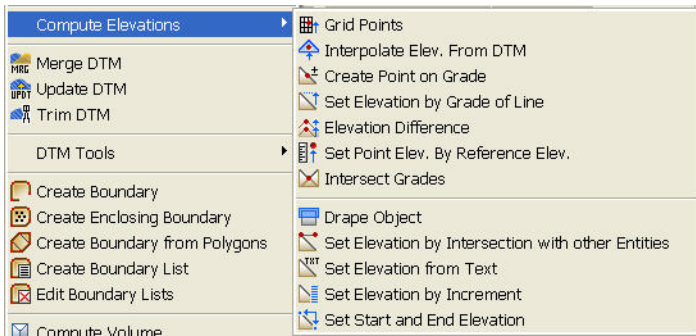


Figure 14-18. Compute Elevation Menu Options

In This Section:

- “Grid Points” on page 14-16
- “Interpolate Point Elevations” on page 14-17
- “Create Point by Grade” on page 14-19
- “Set Elevation by Grade of Line” on page 14-20
- “Elevation Difference” on page 14-21

- “Set Point Elevation by Reference Elevation” on page 14-22
- “Intersect Grades” on page 14-24
- “Drape Object” on page 14-25
- “Set Elevation by Intersection with Other Entities” on page 14-25
- “Set Elevation From Text” on page 14-26
- “Set Elevation by Increment” on page 14-27
- “Set Start and End Elevation” on page 14-28

Grid Points

Use the Grid Points command to create a grid of points with elevations across a project. The levels are interpolated from the current DTM if this exists for the grid area.

To use the Grid Points command, do the following:

1. Click **Surface ▶ Compute Elevations ▶ Grid Points**.

The input bar appears on the lower-left side of the dialog box (Figure 14-19).



Figure 14-19. Grid Point Input Bar

2. Click on a point to serve as the grid origin, or enter either a point number or coordinates into the input bar. The grid will be defined from this point.

The *Create Grid Points* dialog box appears (Figure 14-20).

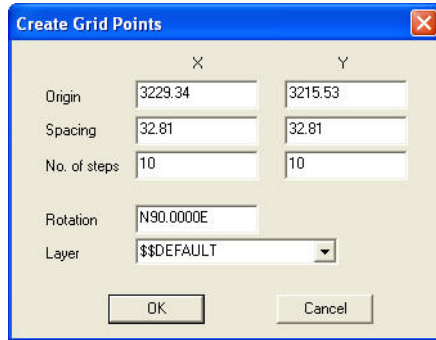


Figure 14-20. Create Grid Points Dialog Box

3. Modify the properties as required, and then click **OK** to create the grid points. For a dialog box description, see Table 14-5 below. Levels will be interpolated from the current DTM surface, where a grid point lies within the extents of the triangular mesh.

Table 14-5. Grid Point Dialog Box Description

Field Name	Field Name Description
Origin	Enter the X and Y coordinates for the start point.
Spacing	Enter the X and Y spacings between the grid points.
No of Steps	Enter the number of grid points created in the X and Y directions.
Rotation	Enter the rotation for the X and Y axis. The default of 90.0000 will create points north and east of each other.
Layer	Select the layer for the grid points.

Interpolate Point Elevations

Use the Interpolate Point Elevations command to interpolate elevations for either a selection of points within the project or for the points that define a boundary from the current triangular surface.

Before interpolating elevations for points, make sure the appropriate DTM is the current surface and, if necessary, select the points for which elevations you want computed.

To use the Interpolate Elevations From DTM command, do the following:

1. Click **Surface** ► **Compute Elevations** ► **Interpolate Elevations from DTM**.

The *Interpolate Point Heights* dialog box appears (Figure 14-21).

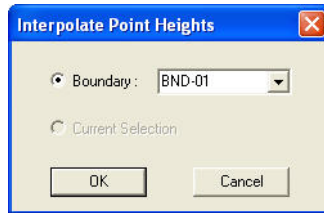


Figure 14-21. Interpolate Point Heights

2. If no points were selected before clicking this option, then choose a boundary from the drop-down menu. The points on the boundary will have levels allocated.

If a selection was made before clicking this option, then the **Current Selection** radio button becomes available.

3. Click **OK** to save.

Levels will be interpolated from the current DTM surface, where a point lies within the extents of the triangular mesh. A report listing the points and their new levels is displayed in a new window in the Report View (Figure 14-22).

Point	East	North	Elev	Prev Elev	Code	Layer
33	927.143	5024.881	50.760	50.760	BLD	BUILD
34	945.432	5031.715	51.060	51.060	BLD	BUILD
35	942.274	5040.166	51.050	51.050	BLD	BUILD
36	934.823	5037.382	51.040	51.040	BLD	BUILD
37	932.174	5044.471	51.040	51.040	BLD	BUILD
38	921.335	5040.421	50.920	50.920	BLD	BUILD

Figure 14-22. Interpolate Height Report

Create Point by Grade

Use the Create Point by Grade command to create a new point on the grade line between two existing points at a nominated elevation. The two existing points must have elevations defined.

To use the Create Point by Grade command, do the following:


1. Click **Surface ▶ Compute Elevations ▶ Create Point on Grade**.

The input bar appears on the lower left of the display area (Figure 14-23).



Figure 14-23. Create Point by Grade Input Bar

2. Do one of the following:
 - Click to select a line that joins the two points to define the grade.
 - Click to select the first point to define the grade.
 - Enter the point number in the input bar.
3. Click to select the second point to define the grade or enter the point number in the input bar.
4. Enter the elevation of the new point into the **Height** field

 on the input bar, or select a reference point of a known elevation.

The grade between the two original points is used to compute the position of a new point. This point has the nominated elevation and will be located on the grade line (Figure 14-24).

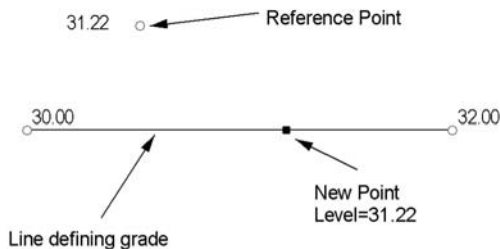


Figure 14-24. Point Created by Grade

Set Elevation by Grade of Line

Use the Set Elevation by Grade of Line command to set elevations on points by the grade of a nominated line that is used to define the slope of a plane. Select additional points to be shifted vertically onto this defined plane (Figure 14-25). The grade of the plane is defined by two existing points with elevations or an existing line joining points with elevations.

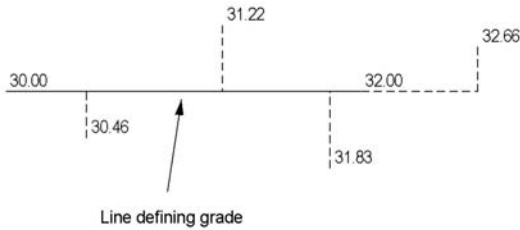


Figure 14-25. Elevations Allocated to Points on a Plane Defined by Grade of Line

To use the Set Elevation by Grade of Line command, do the following:

1. Click **Surface** ► **Compute Elevations** ► **Set Elevation by Grade of Line**.
2. Select one of the following:
 - A start point and then a second point to define the grade line.
 - An existing line to define the grade line.
3. If a point is selected, then select a second point or enter the point number or coordinates into the input bar (Figure 14-26).



Figure 14-26. Set Elevation by Grade of Line Input Bar

If a line is selected, then click a point to set the elevation. The computation is equivalent to moving the point selected perpendicularly on the grade line and applying that height.

4. Continue to select the points whose height is to be set, and then right-click and select **Cancel** from the shortcut menu.

Elevation Difference

Use the Elevation Difference command to vertically compare two surfaces. Using the X,Y positions of points in either or both surfaces, new points are generated with levels that equal the difference in elevation between the surfaces.

If a new layer is not used for the elevation difference points, then the points in the selected layer will have the existing elevations replaced with the height difference levels, so it is recommended to create a new layer for the computed points.

To use the Elevation Difference command, do the following:

1. Click **Surface ▶ Compute Elevations ▶ Elevation Difference**.
The *Height Difference* dialog box appears (Figure 14-27).

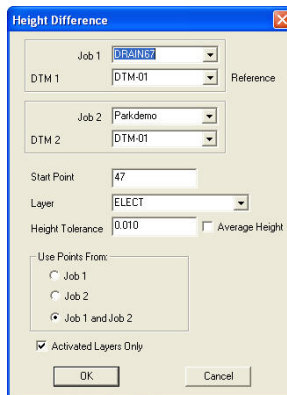


Figure 14-27. Height Difference Dialog Box

2. Modify the properties as required, and then click **OK** to create new points. Table 14-6 describes the height difference fields.

Table 14-6. Height Difference Field Descriptions

Field Name	Field Name Description
DTM 1	This surface is the reference from which the height differences are calculated.
DTM 2	Use this surface for height comparison.
Start Point	The next point in the MAGNET Office project is displayed. Modify this to a larger number if required.

Table 14-6. Height Difference Field Descriptions (Continued)

Field Name	Field Name Description
Layer	Select a new, empty layer to hold the new points. If an existing layer is used, points in the layer will have their levels edited to the height difference.
Height Tolerance	If the height difference is less than the tolerance specified, then the average height can be used.
Average Height	This will average point heights if the height difference is less than the tolerance specified.
Use Points From	Select one of the following options: <ul style="list-style-type: none"> • Job 1 – New points are created with the same X, Y coordinates as those in Job 1. Each point will have a height, which is the difference between the surfaces at this point position. • Job 2 – New points are created with the same X, Y coordinates as those in Job 2. Each point will have a height, which is the difference between the surfaces at this point position. • Job 1 and Job 2 – New points will be created with the same X, Y coordinates as those in both Job 1 and 2. Each point will have a height, which is the difference between the surfaces at this point position.

Set Point Elevation by Reference Elevation

To use the Set Point Elevation by Reference Elevation command, do the following:

1. Click **Surface ▶ Compute Elevations ▶ Set Point Elev. By Reference Elev.**

The prompt line, “Select a point”, displays at the bottom of the screen.

2. After selecting a point, the prompt line, “Input slope, then press Enter”, displays. Input a slope as prompted (Figure 14-28).



Figure 14-28. Input Slope

The *Edit Pt.* dialog box appears (Figure 14-29).

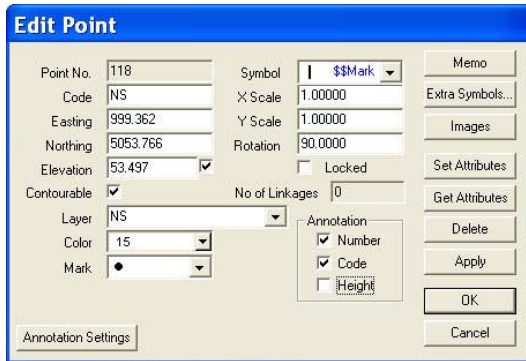


Figure 14-29. Edit the Point

Table 14-7 describes the Set Point Elevation by Reference Elevation fields.

Table 14-7. Set Point Elevation by Reference Elevation Field Description

Field Type	Field Description
Point	The point number selected to edit.
Code	Enter a code name
Easting/Northing	Enter Easting and Northing coordinates.
Elevation	Enter the elevation of the point.
Contourable	Check mark this box to set the point to contourable to be used in DTM creation.
Layer	Select a layer from the drop-down list.
Color	Select a point color from the color library for the point being created.
Mark	Select a symbol to represent the point.
Symbol	Select a symbol type from the drop-down list.
X/Y Scale	Enter the X/Y coordinates for the point.
Rotation	Enter the rotation angle for the point.
Locked	Check mark this box to lock point in place.
No. of Linkages	not editable.
Annotation	Select either Number, Code, or Height to activate the Annotation Settings button.

Table 14-7. Set Point Elevation by Reference Elevation Field Description

Field Type	Field Description
Annotation Settings	Press to change attributes for Point Name, Point Code, or Point Height.
Memo	Press to enter any notes you might have regarding editing of the point.
Extra Symbols	Press to use extra symbols.
Images	Press to insert any images attached to the point.
Set Attributes	Press to set the point attributes to the current project settings.
Get Attributes	Press to get attributes from previous projects.
Delete	Press to delete the point.
Apply	Press to apply the changes made.
OK	Press to save.
Cancel	Press to quit the routine.

Intersect Grades

Use the Intersect Grades command to create a new point between two existing points at a nominated elevation. The two existing points must have defined elevations.

To use the Intersect Grades command, do the following:

1. Click **Surface ▶ Compute Elevations ▶ Intersect Grades**.

The input bar displays.

2. Enter or select the first point, as stated in the prompt line in the left corner of the window.
3. Enter the slope in the input bar, and then press **Enter**.
4. Enter or select the second point.
5. Enter the slope in the input bar, and then press **Enter**.

The grade between the two original points is used to compute the position of the new point. This point has the nominated elevation and will be located on the grade line.

Drape Object

The Drape Object command allows you to interpolate heights from a specified DTM and create points for any selected polyline.

To interpolate heights from a specified DTM, do the following:

Click **Surface** ► **Compute Elevations** ► **Drape Object**.

The prompt line, “*Select a string or polyline*”, displays at the bottom of the screen.

You are prompted to select any existing DTM in the job to interpolate heights from and to select a layer for the new points to be created into, if a polyline is selected.

The *DTM and Layer Selection* dialog box appears (Figure 14-30).

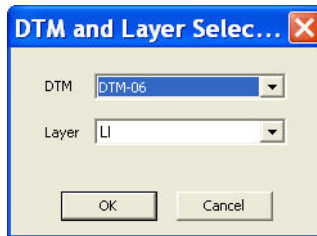


Figure 14-30. Drape Object Menu Option for Height Interpolation

Strings may also be selected to interpolate heights for, however, the string points will remain in its original layer.

Points with heights will be generated based on intersecting points and wherever the strings or polylines intersect with a triangle side.

Set Elevation by Intersection with Other Entities

The Set Elevation by Intersection with Other Entities command sets the elevation of the selected entities by using the intersection points of entities with elevation to calculate the elevation. New points will be created at these intersection points with the elevation of the point set to the calculated elevation.

Imagine a closed square drawn over top of contour lines with elevation. You would use this command to raise the square to the elevation of the contours that its lines intersect with.

Set Elevation From Text

The Set Elevation From Text command will set heights to Points and Polylines based on a selected numerical text.

To set elevation from text, do the following:

- Click **Surface** ▶ **Compute Elevations** ▶ **Set Elevations from Text**.
- The prompt, “*Select Text*”, displays at bottom-left of screen. If the entity selected is not a Point or a POLYline, the linked points will be used. Alternatively, if a selection is made prior to running the option, it will only run the routine once.
- Select the desired text. Right-click with the mouse. The prompt, “Set a Base Elevation”, displays at bottom-left of screen.
- Click on “Set a Base Elevation” to display the *Topcon MAGNET Office* dialog box (Figure 14-31).

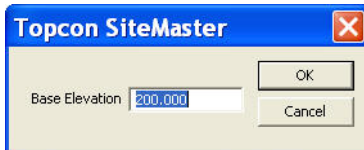


Figure 14-31. Set Elevation From Text



Any text selected which includes alpha characters that are selected will be read without alpha characters.

The last prompt after text selection reads, “Select an Object – ESC to End”.

- Press **ESC** to exit the routine.

Set Elevation by Increment

The Set Elevation by Increment command sets the elevation of selected entities by specifying a base elevation and then an increment to change each object's 'elevation by' value.

To set elevation by increment, do the following:

- Click **Surface ▶ Compute Elevations ▶ Set Elevation by Increment**.

The prompt, "Locate start point" and the **Assign Elevation Input Bar**, displays at bottom-left of screen (Figure 14-32).

Number	<input type="text"/>	East	<input type="text" value="936.681"/>	<input checked="" type="checkbox"/>	North	<input type="text" value="4926.627"/>	<input checked="" type="checkbox"/>
--------	----------------------	------	--------------------------------------	-------------------------------------	-------	---------------------------------------	-------------------------------------

Figure 14-32. The Assign Elevations Input Bar

- Select the starting point for a selection line, then stretch the line until it crosses the objects to set the elevation on (Figure 14-33).

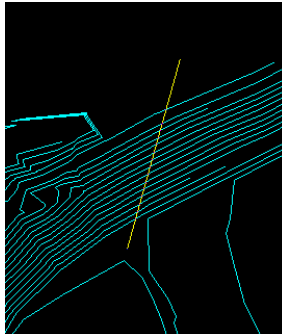


Figure 14-33. Set Elevation by Increment – Results

- When all objects are selected, press **OK**.
- Click **Surface ▶ Compute Elevations ▶ Set Elevation by Increment** again. The *Assign Elevations* dialog box appears (Figure 14-34 on page 14-28).
- Press **ESC** to return to the main screen.



Figure 14-34. Set Elevation by Increment

Set Start and End Elevation

The Set Start and End Elevation command calculates the elevation of points along a polyline or string by specifying an elevation for the starting and ending point elevations.

To calculate the elevation of points along a polyline or string, do the following:

- Click **Surface** ▶ **Compute Elevations** ▶ **Set Start and End Elevation**.

The prompt, “Select a String or Polyline”, appears at bottom-left of screen. Intermediate points along the polyline and/or string will be calculated (Figure 14-35).

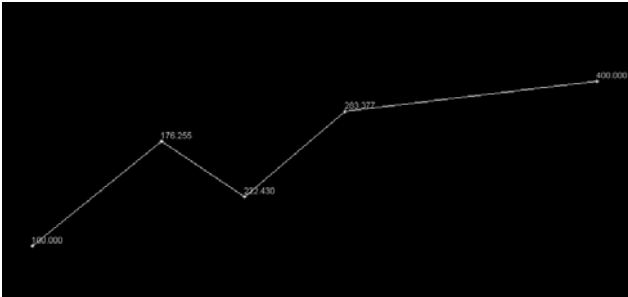


Figure 14-35. Set Start End Elevation – Results

- Select a string or a polyline. The *Assign Elevations* dialog box appears (Figure 14-36 on page 14-29).
- Enter a value for in the Start/End Elevation fields, then press **OK**.
- Press **ESC** to return to the main screen.



Figure 14-36. Enter Start and End Elevation Parameters

Merge DTM

Use the Merge DTM command to create a new DTM surface by merging one or more DTM surfaces into one. This command allows you to merge surface models from other projects into the existing project. The other project must be open in MAGNET Office to allow access to the DTM surfaces.

To use the Merge DTM command, do the following:

1. Click **Surface** ► **Merge DTM**.

The *Merge DTM* dialog box appears (Figure 14-37).

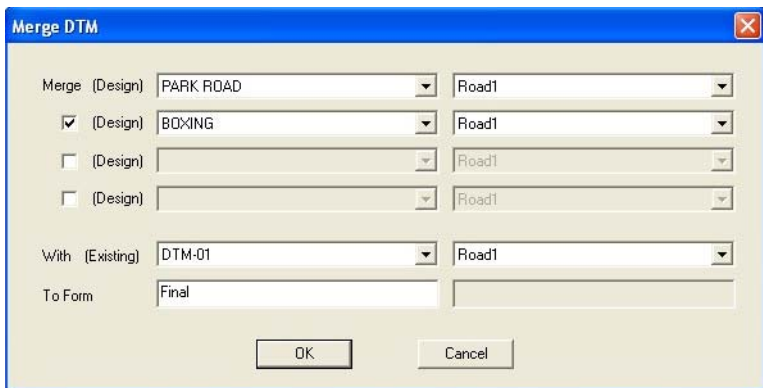


Figure 14-37. Merge DTM

2. From the *Merge (Design)* drop-down menu, select the DTMs to be merged and the project that the DTMs are a part of.

If the DTMs are contained in the active project, the project field need not be edited as the active project will be listed by default.

3. From the ***With (Existing)*** drop-down menu, select the DTM for the others to be merged into in this field.

This DTM should be the largest of all the DTMs to be merged. This DTM will be the Base, and all other DTMS will be used to create a new base. This DTM can also be select from another project. A prerequisite to this is for each project containing a DTM to be merged to be open.

4. In the ***ToForm*** field, enter the name of the final DTM.
The resulting merged DTM will be created in the active project with the mesh and contours active.
5. Click **OK** to create the merged surface.

Additional Information

- To select a DTM from another project, open the project so it is available on the drop-down menu. A DTM cannot be used unless the project containing it is open in MAGNET Office.
- The triangles displayed in the first or “design” DTM will replace the triangles in the equivalent coordinate positions in the second or existing DTM, using the outside edge of the first DTM as a slope line.
- If the edge of the first DTM does not create a slope line on the surface with the second DTM, there will be an interpolation of heights between the two surfaces, which may not be what you expect.

Update DTM

Use the Update DTM command to update the current DTM surface after editing the elevation values of a point or points that are already used for the DTM. The contours display is updated.

To use this command, click **Surface ▶ Update DTM**. The DTM is updated, and the contour display is modified to reflect the changes.

Trim DTM

To remove unwanted triangles inside or outside a boundary, do the following:

1. Click **Surface** ► **Trim DTM**.
2. Select a boundary, as stated in the prompt line.

The *Trim DTM* dialog box appears (Figure 14-38).

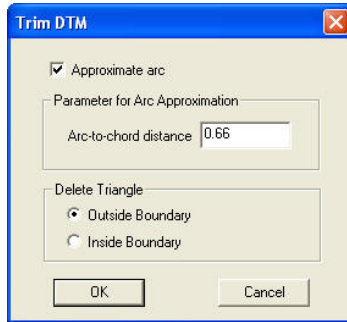


Figure 14-38. Trim DTM Dialog Box

3. If the boundary contains arcs, select the **Approximate arc** check box to approximate the arcs with chord segments by and enter the **Arc-to-chord distance**.
4. Choose to delete triangles inside or outside of the boundary, and then click **OK**.

The triangles only appear on the inside or outside of the boundary, depending on your selection.

DTM Tools

The Digital Terrain Model (DTM) menu contains several tools for creating, editing, and displaying triangular surface model and contours (Figure 14-39 on page 14-32). The surface model is a triangular irregular network (TIN) and displays on the screen as a triangular mesh with a dot at the centroid of each triangle.

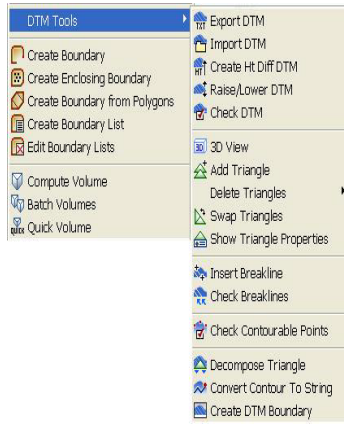


Figure 14-39. Surface Menu – DTM Tools

Any lines defined as breaklines may be inserted into the triangular model as it is created to ensure a change of grade occurs at this line.

The contours displayed from this DTM are user-defined with the option of setting the intervals for the major and minor contours, colors, and line styles.

You can create multiple DTM surfaces. Each surface is allocated a default name that you can edit before the DTM is created

In This Section:

- “Export DTM” on page 14-33
- “Import DTM” on page 14-33
- “Create Height Difference DTM” on page 14-34
- “Raise/Lower DTM” on page 14-35
- “Check DTM” on page 14-36
- “3DViewer Screen” on page 14-40
- “Add Triangle” on page 14-45
- “Delete Triangle” on page 14-46
- “Swap Triangles” on page 14-52
- “Show Triangle Properties” on page 14-52

- “Insert Breakline” on page 14-53
- “Check Breaklines” on page 14-56
- “Check Contourable Points” on page 14-58
- “Decompose Triangles” on page 14-59
- “Convert Contour to String” on page 14-61
- “Create DTM Boundary” on page 14-63

Export DTM

Use the Export DTM command to export a specified surface DTM to a Topcon TN3 file for use with 3DMC.

To export a surface DTM file, do the following:

1. Click **Surface** ▶ **DTM Tools** ▶ **Export DTM**.
The *Save As* dialog box appears.
2. Locate the folder in which you want to save the file, and click **Save**.

Import DTM

Use the Import DTM command to import a surface DTM from another project into the current project.

To import a surface DTM file, do the following:

1. Click **Surface** ▶ **DTM Tools** ▶ **Import DTM**.
The *Import DTM* dialog box appears (Figure 14-40).

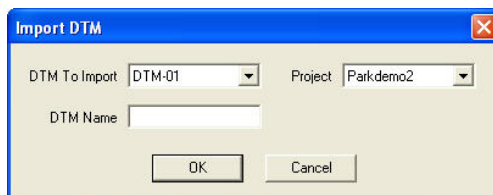


Figure 14-40. Import DTM Window

2. Select a **DTM To Import** and a **Project** from the drop-down list.
3. Enter a **DTM Name**, then click **OK**.

The DTM is imported into the selected project.

Create Height Difference DTM

Use the Create Ht Diff DTM command to create a cut/fill color map.

1. Click **Surface** ► **DTM Tools** ► **Create Ht Diff DTM**. The *Create Height Difference DTM* dialog box displays (Figure 14-41).
2. On the *Create Height Difference DTM* dialog box (Figure 14-41), select or enter the following:
 - First DTM – select for a calculation of height difference; usually the existing surface, or before design, DTM.
 - Second DTM – select for a calculation of depth difference; usually the design DTM. The cut/fill depths will be calculated between these two specified surfaces.
 - From Level – check mark this box to nominate a DTM and then to specify a particular elevation or level to calculate the depth difference between them.
 - Output DTM – must enter an output value to create a height difference.
 - Cut/Fill DTM – click on the down arrow to display the *Basic Colors* screen to select a cut/fill color.

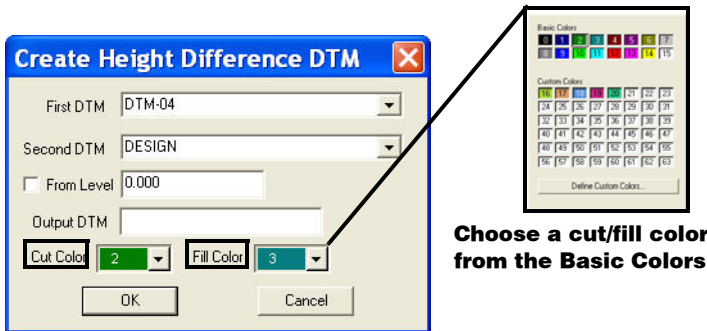


Figure 14-41. Create Height Difference DTM Dialog Box

Raise/Lower DTM

Use the raise/lower DTM command to raise or lower a specified surface(s) vertically by a defined amount (Figure 14-42).

1. Click **Surface** ► **DTM Tools** ► **Raise/Lower DTM**. The *Raise/Lower DTM* dialog box displays (Figure 14-42).
2. Table 14-8 describes the *Raise/Lower DTM* dialog box field options.

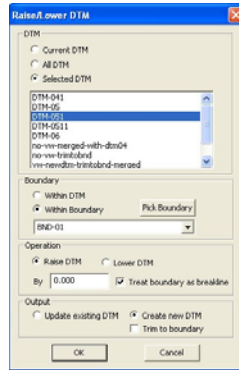


Figure 14-42. Raise/Lower DTM Dialog Box

Table 14-8. Raise/Lower Boundary DTM Field Descriptions


Field	Description
DTM	<ul style="list-style-type: none"> • Current DTM: Enable to apply the specified vertical adjustment to the DTM currently set for the project. To verify the current DTM, press the DTM hot button  (green triangle), on the Status Bar located at the bottom of the screen. • All DTM: Enable to apply vertical adjustments to all DTMs in the project. • Selected DTM: Enable to apply vertical adjustment to only the DTMs selected from the DTM window below. Multiple DTMs can be selected for adjustment.

Table 14-8. Raise/Lower Boundary DTM Field Descriptions (Continued)

Field	Description
Boundary	<ul style="list-style-type: none"> • Within DTM: Enable to raise or lower the selected DTM(s) within the boundary established when creating the DTM from the drop-down list. NOTE: The entire DTM will be encompassed. • Within Boundary: Enable to raise or lower the selected DTM(s) within a selected boundary from the drop-down list. • Pick Boundary: Enable to allow the selection of a boundary from the survey view (main drawing screen) from the drop-down list.
Operation	<ul style="list-style-type: none"> • Raise DTM: Enable to raise the specified DTM(s) vertically by the value entered in the <i>By</i> field below. • Lower DTM: Enable to lower the specified DTM(s) vertically by the value entered in the <i>By</i> field below. • Treat boundary as breakline: Check mark this box to assist in crating vertical walls resulting from the elevation change of the surface when creating a new DTM or when updating an existing DTM.
Output	<ul style="list-style-type: none"> • Update existing DTM: Enable to update the existing DTM(s) with specified changes defined. NOTE: There will be an overwrite of the original DTM. • Create new DTM: Enable to create a new DTM that reflects the specified changes in the elevation of the DTM(s). The new DTM will automatically be named using the DTM already specified for raising or lowering, followed by an appending suffix of 1. • Trim to Boundary: Check mark this box to trim the modified DTM to the specified boundary.

Check DTM

Use the Check DTM command to check the connectivity of the triangles in the DTM. It will highlight where the overlapping triangles (if any) are located and save them in the layer you selected. You can also check the boundaries (exterior and interior) of the DTM and report a self-intersecting boundary. Use this command to check an imported DTM before using it in a project.

Click **Surface** ► **DTM Tools** ► **Check DTM**. The *DTM Check* dialog box appears (Figure 14-43).

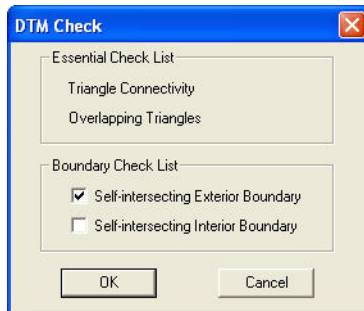


Figure 14-43. DTM Check Window

3D View

3D views are representative of the surface created from a digital terrain model in Survey View.

To view a file in the 3D viewer, do the following:

1. Click **Surface** ► **DTM Tools** ► **3D View**.



Be sure the layers on which the alignments are defined are active in Survey View.

The *3D View* dialog box appears (Figure 14-44 on page 14-38).

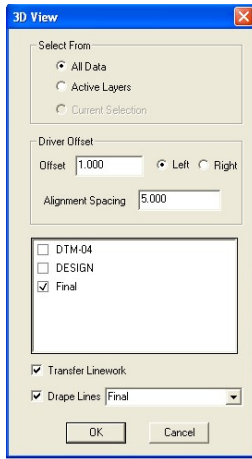


Figure 14-44. 3D Viewer Dialog Box

Table 14-9 describes the parameters for the surface in the *3D View* dialog box (Figure 14-44).

Table 14-9. 3DViewer Parameters Field Descriptions

Field Name	Field Description
Select From	<ul style="list-style-type: none"> • All Data – displays all drawing data in the 3D Viewer, whether the layer visibility is on or off. In addition, the ‘transfer linework’ option below must be check marked in order to display the linework. • Active Layers – displays only the data contained in the active layers within the 3D Viewer. In order to display linework, the ‘transfer linework’ option below must be check marked. • Current Selection – displays only the entities within the current selection (when available). In order to display linework, the ‘transfer linework’ option below must be check marked.
Driver Offset	This option is used for the inspection of the surface at a specified perpendicular offset; select either Left or Right of the alignment.
Alignment Spacing	Enter a value to produce a detailed drawing of the alignment. The lower the value, the smoother the curvature will be.

Table 14-9. 3DViewer Parameters Field Descriptions (Continued)

Field Name	Field Description (Continued)
DTM Window	Displays all available DTMs in the project. Multiple DTMs can display simultaneously. If the difference in the surfaces is the desired visual effect, you should create DTMs with different colors.
Transfer Linework	Check mark this box to toggle on/off the ability to transfer specific linework in the 3D Viewer.
Drape Lines	Check mark this option to drape all selected lines onto the selected surface in the 3DViewer.
OK	Press to see the specified surface in 3D (Figure 14-45 on page 14-39).
Cancel	Press to quit and return to the main screen.

Figure 14-45 shows how the surface view will display in 3D view.

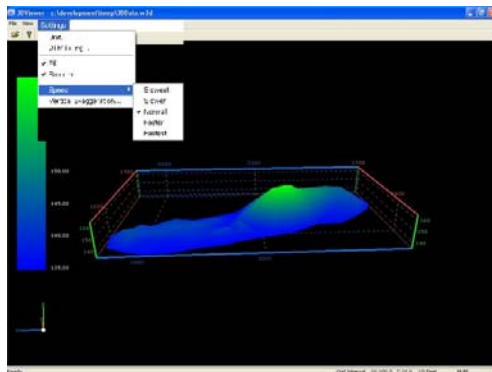


Figure 14-45. Surface View Shown in the 3DViewer

- To select a position in which to view the data, press and hold the left mouse button. Move the pointer to a new position (the 3D image moves with the pointer) and release the left mouse button.
- Click **Help ► About** on the *3DViewer* dialog box to use the various keyboard functions listed in Figure 14-46 to move around the 3DViewer and to select a position to view the data.

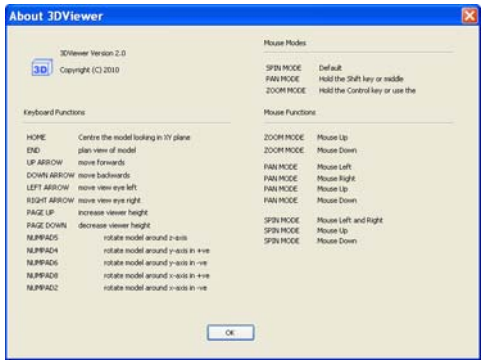


Figure 14-46. About 3D Viewer

3DViewer Screen

The **3DViewer** screen displays a 3D visualization (Figure 14-47) of all digital terrain models that you can view from any selected position.

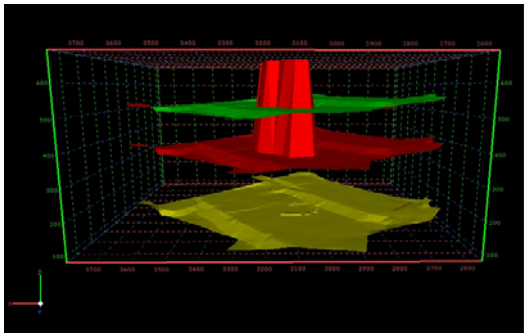


Figure 14-47. 3D Visualization

View Menu Options On the **3DViewer** screen display (Figure 14-48 on page 14-41), click on the **View** menu option to set the required parameters for the view.

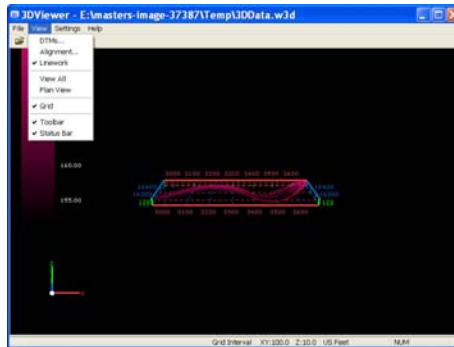


Figure 14-48. 3D Viewer – View Menu

- DTMs – press to toggle on/off the display of available DTMs within the 3DViewer.
- Alignment – select an Alignment file, then select either **Forward** or **Reverse** for *Direction of Travel* and enter a distance in the *Driving Level* field (Figure 14-49).

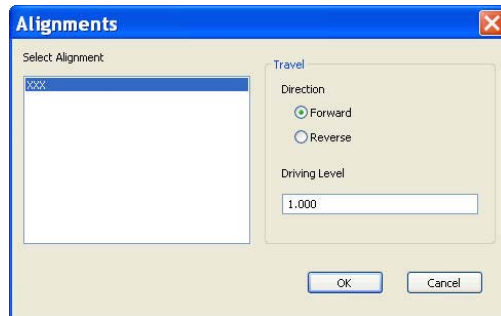


Figure 14-49. The Alignments Dialog Box

- Linework – select this option to display linework on the 3DViewer screen display (Figure 14-50 on page 14-42).

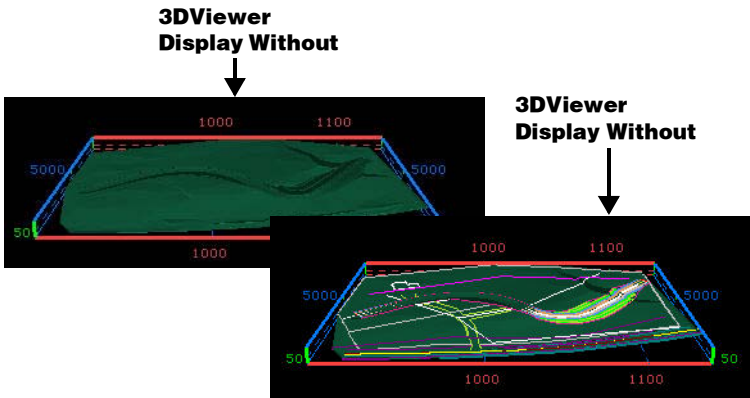


Figure 14-50. Display Linework on the 3DViewer Screen Display

- View All – zooms to the extent of the 3D view.
- Plan All – select to change to plan view of the 3DViewer screen display.
- Grid – check mark this option to display the grid on the 3DViewer screen display (Figure 14-51).

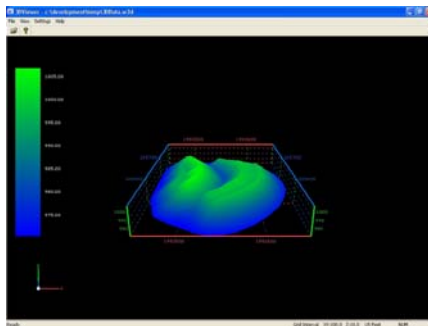


Figure 14-51. Grid Display in 3D Image

- Toolbar – toggle on/off to show/hide the toolbar.
- Status Bar – toggle on/off to show/hide the status bar.

Settings Menu Options The 3DViewer Settings menu option sets parameters for unit of measurement, surface display, DTM mesh, triangular mesh, zooming/panning speed, and vertical exaggeration value of the 3D view (Figure 14-52 on page 14-43).

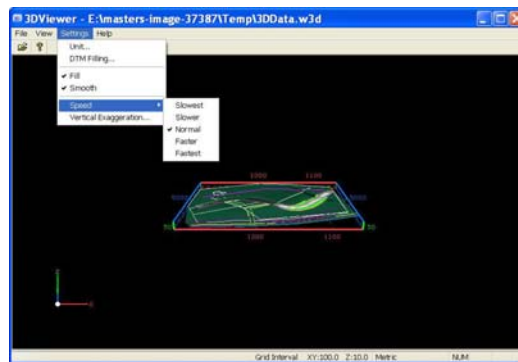


Figure 14-52. 3DViewer Settings Menu Options

The Settings menu has the following options (Figure 14-52):

- Unit – select either *Metric*, *US Feet*, or *International Feet* as the unit of measurement.

When the units are changed, the grid will be updated to display the selected unit of measurement.

- DTM Filling – this option controls how the surface displays. The *DTM Fill* dialog box displays (Figure 14-53).
 - Based on light source: enable this option to display the colored fill of the surface model triangles.
 - Based on elevations: enable this option to provide a color zone elevation display with the lowest and highest colors based on user settings.
 - Color of lowest point: select a color for the lowest elevation on the surface.
 - Color of highest: select a color for the highest elevation on the surface.

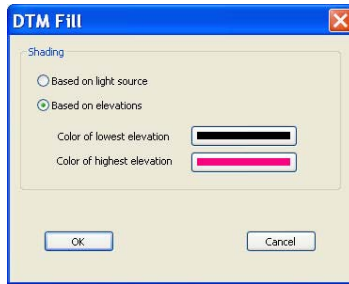


Figure 14-53. DTM Fill Dialog Box

- Fill – check mark this option to control the filling of the DTM mesh (Figure 14-53). If check marked, the mesh will be filled, based upon the DTM filling settings. If unchecked, only the mesh will display.

- Smooth – smooths the triangular mesh lines to provide a smoother appearance (Figure 14-53). The smoothing for the color transition of the mesh is also controlled.

NOTE: This setting should typically be left on to obtain the best visual effect for the surface.

- Speed – controls the speed of zooming and panning while in 3D view. Select either *Slowest*, *Slower*, *Normal*, *Faster*, or *Fastest* (Figure 14-53).

In addition, when following an alignment along the surface, this setting controls the speed of the “drive”. This speed setting is available at any time while in the 3DViewer in the context menu with a right-click of the mouse.

- Vertical Exaggeration – sets the value of the vertical exaggeration of the 3D view (Figure 14-53).

This is useful with the display of flat surfaces with very little relief. Setting this value to a higher number will show the relief in an exaggerated view.

About 3DViewer

To get help navigating the 3DViewer, click **Help ► About 3DViewer**. The *About 3DViewer* screen (Figure 14-54) appears to display the version number and copyright date for your information. Keyboard Function, Mouse Modes, and Mouse Functions gives helpful information about how to move around on screen.



Figure 14-54. About 3DViewer – 3D View Information

Add Triangle

Use the Add Triangle command to manually add triangles to the current DTM triangular mesh. This may be necessary to replace deleted triangles or to include a new contourable point (Figure 14-55).

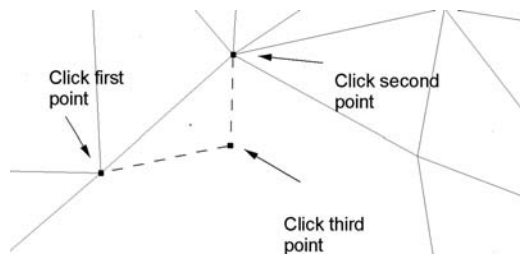


Figure 14-55. Net Triangle Added from Three Existing Points with Elevations

To manually add triangles to the current DTM triangular mesh, do the following:

1. Click **Surface ▶ DTM Tools ▶ Add Triangle**.
A confirmation message may appear (Figure 14-56).
2. Click **Yes** to turn on the current DTM.

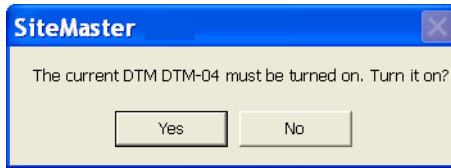


Figure 14-56. DTM Turned On Confirmation

3. Select the first point. Preferably, this should be at the vertex of an existing triangle that is adjacent to the area where the new triangle will be formed.
4. Select the second point. Preferably, this should be at the vertex of an existing triangle that is adjacent to the area where the new triangle will be formed.
5. Select the third point to form the third vertex of the new triangle. This point should have an elevation and be set as contourable. The new triangle is created, and a dot will display at the centroid of the triangle.
6. When you have finished, press **Esc** or select **Cancel** from the right-click shortcut menu.



The vertex of some triangles will be a “hidden” point created during the insertion of breaklines. These points cannot be selected during the add triangle routine.

Delete Triangle

Use the Delete Triangle command to delete one triangle at a time (manually), delete a triangle by Line, by Boundary, or by Length and Angle.

NOTE: The DTM must display on the screen before using this command (Figure 14-57).

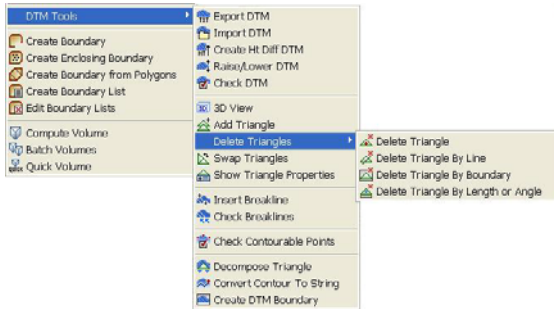


Figure 14-57. DTM Tools – Delete Triangle Options

In This Section:

- “Delete a Triangle”
- “Delete Triangle By Line” on page 14-47
- “Delete Triangle By Boundary” on page 14-48
- “Delete Triangle By Length or Angle” on page 14-50

Delete a Triangle To manually delete one triangle at a time from the current DTM, do the following (Figure 14-58 on page 14-48):

1. Click **Surface** ▶ **DTM Tools** ▶ **Delete Triangle** ▶ **Delete Triangles**.
2. Click the center of a triangle. The dot at the centroid is deleted, removing the triangle from the DTM.
3. Continue to delete triangles as necessary.



If a triangle is deleted in error, select **Undo** to restore it.

4. Right-click on the mouse and Press **Esc** or select **Cancel** from the menu to end the command. The contours are now regenerated. See also “Delete Triangle By Line” on page 14-47.

Delete Triangle By Line Use the Delete Triangle by Line command to draw a string, comprising one or more segments, across triangles

that you want to delete from the current DTM. The DTM must be displayed on the screen before using this command.

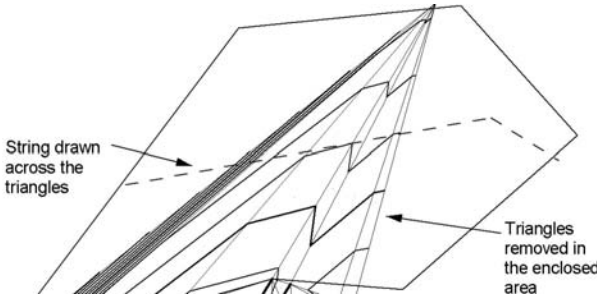


Figure 14-58. All Triangles Crossed by the String Line are Deleted

To delete triangles by line, do the following:

1. Click **Surface ▶ DTM Tools ▶ Delete Triangle ▶ Delete Triangle By Line**.
2. Click on a point inside or outside of the triangle, and drag the line across all triangles you want to delete.
3. Right-click and select **Insert String** from the shortcut menu. A confirmation message appears (Figure 14-59).



Figure 14-59. Confirmation Message

4. Click **Yes** to delete the highlighted triangles.
5. Right-click on the shortcut menu and press **Esc** or select **Cancel** to end the command.

The contours are regenerated.

Delete Triangle By Boundary Use the Delete Triangle by Boundary command to delete triangles from the current DTM by selecting points around the edge of the proposed triangle, a defined area to create a boundary, or by using an existing boundary (Figure 14-60 on page 14-49). The DTM must display on the screen before using this command.

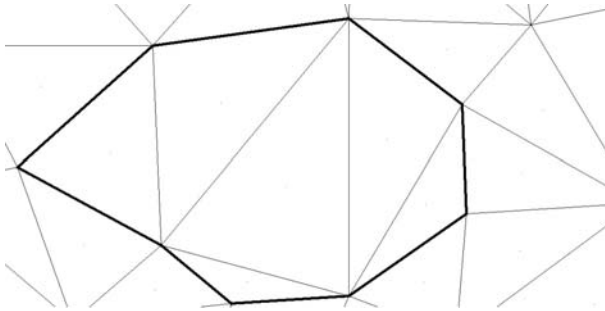


Figure 14-60. Triangles Deleted in the Defined Boundary

To delete triangles by boundary, do the following:

1. Click **Surface** ▶ **DTM Tools** ▶ **Delete Triangle** ▶ **Delete Triangle By Boundary**.

The *Delete Triangles By Boundary* dialog box appears (Figure 14-61)

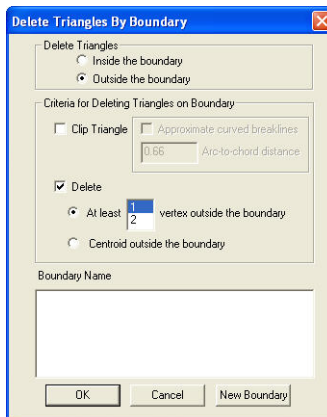


Figure 14-61. Delete Triangles By Boundary

2. Modify the properties as necessary, and then click **OK**. For a description of the *Delete By Boundary* fields, see Table 14-10 below, or click **OK** and select a boundary with the pointer.

Table 14-10. Delete Triangles By Boundary Field Descriptions

Field	Description
Inside Boundary	Deletes triangles inside the defined boundary.

Table 14-10. Delete Triangles By Boundary Field Descriptions (Continued)

Field	Description
Outside Boundary	Deletes triangles outside the defined boundary.
Clip Triangle	Deletes only the triangles that are totally enclosed in the boundary.
Approximate curved breaklines	Creates additional triangles so triangle edges are formed along chords of the curved breaklines. NOTE: This is used if the boundary includes an arc.
Arc-to-chord Distance	Accepts the default value or enter another value.
Delete	Deletes triangles crossing the boundary. If this option is selected, then choose one of the following: At least 1 vertex outside the boundary: Deletes all triangles that cross the boundary that have at least 1 vertex outside the boundary. At least 2 vertex outside the boundary: Deletes all triangles that cross the boundary that have 2 vertices outside the boundary. Centroid outside the boundary: Deletes all triangles that cross the boundary but have the centroid outside the boundary.
Boundary Name	Lists existing boundaries. Select one to use for the deletion of triangles or define a new boundary by clicking New Boundary .
New Boundary	Defines a new boundary. Click on existing points to create a boundary or draw the boundary with new points. The boundary is not saved using this option.

When a boundary is selected, a confirmation message appears (Figure 14-62).



Figure 14-62. Confirmation Message

3. Click **Yes** to delete the highlighted triangles.

Delete Triangle By Length or Angle Use the Delete Triangle by Length or Angle command to delete triangles from the current DTM

by nominating the maximum side length of triangles for the DTM. All triangles with a longer side length than the maximum are deleted. The DTM must be displayed on the screen before using this command.

To use the delete triangle by length or angle command, do the following:

1. Click **Surface ▶ DTM Tools ▶ Delete Triangle ▶ Delete Triangle By Length or Angle**.

The *Delete Triangles* dialog box appears (Figure 14-63).

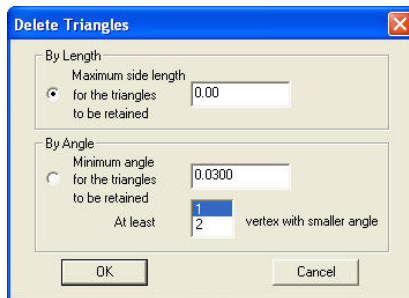


Figure 14-63. Delete Triangles

2. Select either **By Length** or **By Angle** and enter the desired values for the triangle.
3. Click **OK**.

A confirmation message appears (Figure 14-64).

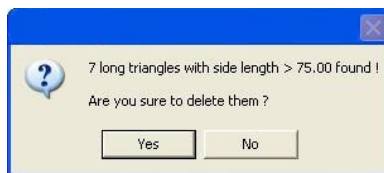


Figure 14-64. Confirmation Message

4. Click **Yes** to delete the highlighted triangles.

Swap Triangles

To swap two adjacent triangles so that the triangles are defined to their opposite diagonals (Figure 14-65), use the Swap Triangles command. This routine modifies the contours.

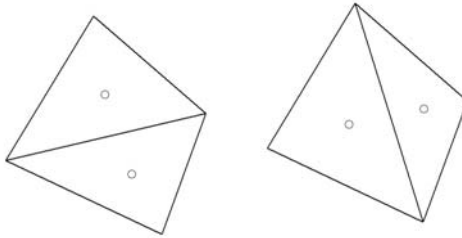


Figure 14-65. Swap Two Triangles with a Common Triangle Side

To swap two adjacent triangles, do the following:

1. Click **Surface** ▶ **DTM Tools** ▶ **Swap Triangles**.
2. Select one triangle with a common triangle edge
3. Select the other triangle.

The common edge of the two triangles is changed to the opposite diagonal.

4. When you have finished, press **Esc** or select **Cancel** from the right-click shortcut menu.

Show Triangle Properties

To view the properties of a triangle, click **Surface** ▶ **DTM Tools** ▶ **Show Triangle Properties**.

The *Triangle Properties* dialog box appears (Figure 14-66 on page 14-53).



Figure 14-66. Triangle Properties Window

Insert Breakline

Use the Insert Breakline command to ensure all of the breaklines on the active layers are inserted as breaklines into the DTM. Any breakline – line or arc – that has not already been used will now be inserted into the current DTM. This enables you to make sure that all breaklines have been inserted in the DTM after you have completed editing lines and arcs (Figure 14-67 and Figure 14-68 on page 14-54).

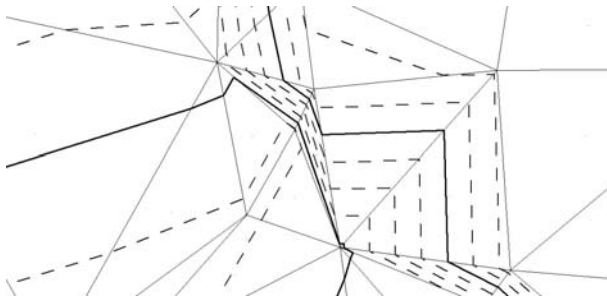


Figure 14-67. Contours and Triangles Before Breaklines are Inserted

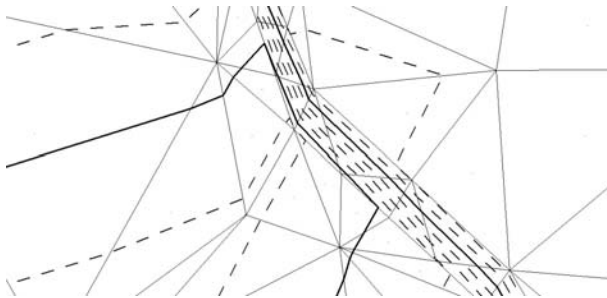


Figure 14-68. Contours and Triangles After Breaklines have been Inserted

To insert breaklines, do the following:

1. Click **Surface** ▶ **DTM Tools** ▶ **Insert Breakline**.

The *Insert Breaklines Into DTM* dialog box appears (Figure 14-69).

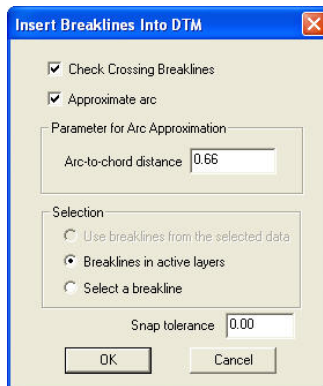


Figure 14-69. Insert Breaklines Into DTM Dialog Box

2. Modify the breakline parameters as necessary, and click **OK**.
Table 14-11 describes the DTM fields.

Table 14-11. Insert Breaklines Into DTM Field Descriptions

Field	Description
Check Crossing Breaklines	Makes sure the intersection is correct.
Approximate arc	Creates breaklines around curves.

Table 14-11. Insert Breaklines Into DTM Field Descriptions (Continued)

Field	Description
Arc-to-Chord Distance	Accept the default value or enter another value. More triangles are formed around the curve when a smaller value is used.
Selection	
Use Breaklines From the Selected Data	This is only available if data is selected before running this command.
Breaklines in Active Layers	Checks all of the breaklines in the active project layers.
Select a Breakline	Checks and inserts one breakline. If you choose this option, follow the prompts on the lower left side of the window.

Selected or Active Data If either **Use breaklines from the selected data** or **Breaklines in active layers** is chosen, then all breaklines in the selection or project on the active layers are checked and the triangles reformed in the current DTM so the triangle edges run along the breaklines and do not cross.

A report displays containing the details of any crossing breaklines, points with no levels at the end of the breaklines, and any other problems (Figure 14-70). The report is similar to that created in the **Create DTM** command.

```

FileMaster -                               Jul 02, 2010 11:30AM Page: 1

Check Breaklines
-----
Line (101, 102) ignored - a point of the breakline is non-contourable.
String (1019, 1016) ignored - it intersects with other breakline.
Line (102, 103) ignored - it intersects with other breakline.
String (1021, 1025) ignored - it intersects with other breakline.
String (1019, 1025) ignored - it intersects with other breakline.
String (1004, 1007) ignored - it intersects with other breakline.
Line (103, 104) ignored - it intersects with other breakline.
String (1018, 1012) ignored - it intersects with other breakline.

Breakline Insertion Report
-----

Summary
-----
59 points used in the triangulation.
Highest point elevation      : 343.209
Lowest point elevation     : 219.924
Total number of triangles in the DTM: 109
Total plan area of the DTM  : 29286.348
Total surface area of the DTM: 29854.315

```

Figure 14-70. DTM Error Report

Check the report to ensure the DTM is accurately created. You may have breakline layers turned on that should not be included in the DTM. Use the “Decompose Triangles” on page 14-59 command to identify any crossing breaklines.

One Selected Breakline If **Select a breakline** was chosen, then select a line that has not already been used as a breakline in the DTM.

The triangles are reformed so the triangle edges run along the breakline and do not cross it.

Add additional breaklines or select **Cancel** from the right-click menu to end the command.

Check Breaklines

Use the Check Breaklines command to highlight all of the breaklines on the active layers. The breaklines are scanned to find any crossing breaklines and any intersection points are identified in a report.

Crossing breaklines can cause an error in the surface model and produce inaccurate contours. Any crossing breakline is ignored by the breakline insertion routines.

Crossing breaklines are usually errors caused by incorrect stringing, field procedure, or mistakes in the code library. Check all crossing breaklines and modify, segment, or delete the lines to ensure the formation of an accurate surface model (Figure 14-71).

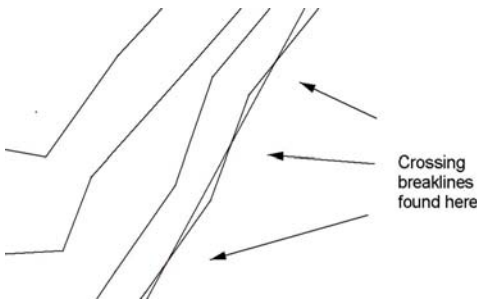


Figure 14-71. Crossing Breaklines Found

To highlight all of the breaklines on active layers, do the following:

1. Click **Surface** ▶ **DTM Tools** ▶ **Check Breaklines**.

The **Check Breaklines** dialog box appears (Figure 14-72 on page 14-57).



Figure 14-72. Check Breaklines Window

2. Make a selection, and click **OK**.

A report appears showing all breaklines or only the invalid breaklines, depending on your selection (Figure 14-73).

```

SiteMaster - Jul 12, 2010 12:12PM Page: 1

Check Breaklines
String (700, 721) - a point of the breakline is non-contourable.
String (764, 795) - a point of the breakline is non-contourable.
String (86, 245) - a point of the breakline has no height.
String (995, 1006) - a point of the breakline is non-contourable.
String (601, 635) - a point of the breakline is non-contourable.
String (88, 245) - it intersects with other breakline.
String (989, 1006) - it intersects with other breakline.
Line (24, 25) - it intersects with other breakline.
Line (15, 16) - it intersects with other breakline.
String (892, 923) - it intersects with other breakline.
Line (60, 21) - it intersects with other breakline.
String (700, 721) - it intersects with other breakline.
String (854, 865) - it intersects with other breakline.
String (524, 955) - it intersects with other breakline.
String (732, 763) - it intersects with other breakline.
String (601, 635) - it intersects with other breakline.
String (668, 699) - it intersects with other breakline.
String (834, 869) - it intersects with other breakline.
String (826, 859) - it intersects with other breakline.
String (796, 827) - it intersects with other breakline.
String (764, 795) - it intersects with other breakline.

```

Figure 14-73. Check Breaklines Report

Additional Information

- Crossing breaklines can occur when the wrong string number is entered for a survey point in the field, so points are allocated to the wrong stringline.
- Crossing breaklines can occur when the point is allocated the wrong feature code.
- Crossing breaklines can occur when surveying the top and base of a vertical or near vertical feature. Make sure that shots at the top and base are in a similar position so the lines do not overlap.

Check Contourable Points

Use the Check Contourable Points command to check for contourable points that have the flag turned on.

To check which points have the contourable flag turned on, do the following:

1. Click **Surface** ► **DTM Tools** ► **Check Contourable Points**.

The *Contourable Points* confirmation box appears (Figure 14-74), asking if you would like to generate a contourable points report.

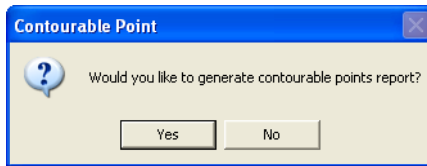


Figure 14-74. Contourable Point Confirmation Screen

2. Press **Yes** to display the **Contourable Point Report** (Figure 14-75).

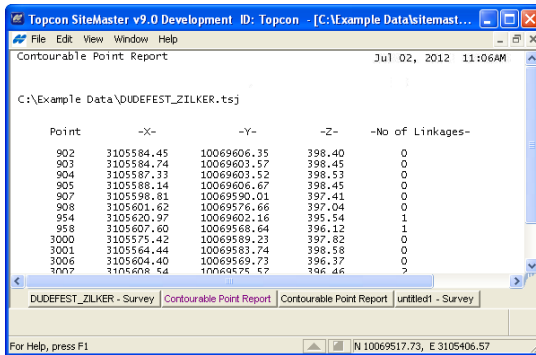


Figure 14-75. The Contourable Point Report

To view the highlighted points when selecting the report option, either close the report or switch back to the Survey View.

- Press **No** to display the points highlighted with a red X as shown in Figure 14-76.

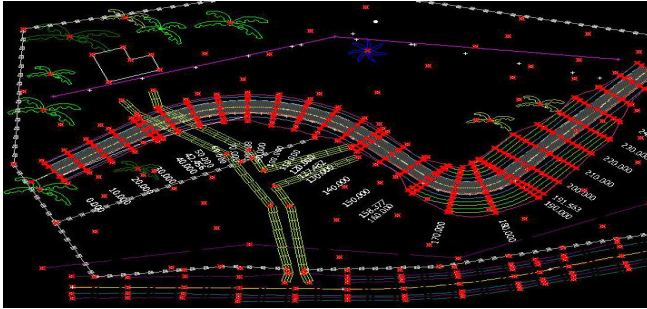


Figure 14-76. Points are Highlighted with a Red X

- Press the ESC key to make the red marks disappear.

Modify the properties as necessary, and then click **OK**. For field descriptions.

Decompose Triangles

Use the Decompose Triangle command to decompose triangles in the current digital terrain model. When a line, arc, polygon, or string is selected, additional triangles are created along the entity as if the entity is a breakline. The purpose is to insert additional triangles that do not alter the existing contours.

The points defining the entity do not need to have levels or be set as contourable. Heights may be allocated to these points at a later date to modify the DTM.

To decompose triangles in the current DTM, do the following:

- Click **Surface** ▶ **DTM Tools** ▶ **Decompose Triangle**.

The *Decompose Triangles* dialog box appears (Figure 14-77 on page 14-60).

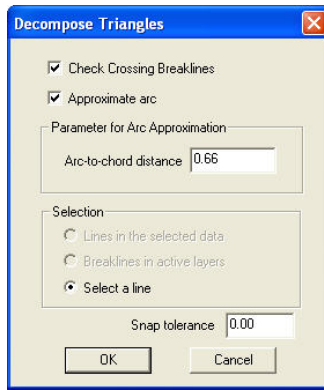


Figure 14-77. Decompose Triangle

2. Modify the properties as needed, and click **OK**. For a description of the fields, see Table 14-12 below.

Table 14-12. Decompose Triangle Field Descriptions

Field	Description
Check Crossing Breaklines	Makes sure the intersection is correct.
Approximate Arc	Creates breaklines around curves.
Arc-To-Chord Distance	Accept the default value or enter another value. More triangles are formed around the curve when a smaller value is used.
Lines in the Selected Data	This option is available if a selection of data is made before using this command. See also “Lines in the Selected Data” on page 14-60.
Breaklines in Active Layers	Checks all of the breaklines in the active project layers.
Select a Line	Enables you to select a line, arc, string, or polygon. If this option is chosen, then follow the prompts on the Status Bar. The triangles are reformed so the edges run along the entity and do not cross it.
Snap Tolerance	The distance of the line to a point before the point is no longer recognized.

Lines in the Selected Data If this option is used, then all of the entities in the selection will be used to create new triangles. The

triangles are reformed in the current DTM, so the triangle edges run along the selected entities and do not cross them. The end points will be at the triangle vertex. The contours are not modified.

A report is displayed, containing the details of any entities that have been inserted into the DTM using this procedure.

Convert Contour to String

Use the Convert Contour To String command to create a selected contour as a stringline. A series of points with the elevation of the contour line are created as nodes on the stringline. A range of contour lines may also be converted to stringlines. The contour strings are useful for defining slope lines or boundaries for such routines as landfill calculations.

To use the Convert Contour To String command, do the following:

1. Click **Surface** ► **DTM Tools** ► **Convert Contour To String**.
2. Select a contour or select a range of contours by right-clicking and choosing **Select Range**.

If you chose **Select Range**, then the *Contour Range* dialog box appears (Figure 14-78).

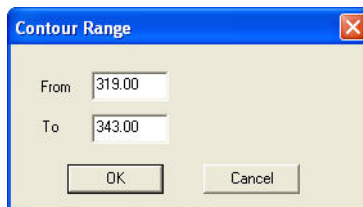


Figure 14-78. Contour Range

Edit the **From** and **To** fields to define the range of contours to be converted to strings, and then click **OK**. The strings are created for the contour range one at a time. The *Create String* dialog box (Figure 14-79 on page 14-62) displays for each contour line.

3. On the **Create String** dialog box (Figure 14-79 on page 14-62), modify the properties as needed, and click **OK**. For a description of the create string fields, see Table 14-13 on page 14-62.

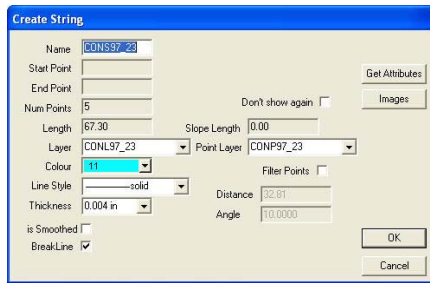


Figure 14-79. Create String Dialog Box

Table 14-13. Create String Field Descriptions

Field	Description
Name	A name is displayed using the format CONSxx_xx. The xx_xx represents the elevation level of the contour string to two decimal places.
Start Point and End Point	This is not used.
Num Points	The number of points created for the contour string.
Length	The length of the contour string.
Layer	A new layer is defined using the format CONLxx_xx. The xx_xx represents the elevation level of the contour string to two decimal places. The string line is written to this layer.
Color	The color is the same one used for the original contour.
Line Style	The line style is the same one used for the original contour.
Thickness	The line thickness is the same one used for the original contour.
Is Smoothed	Applies smoothing to the new string line.
Breakline	Defines the new string line as a breakline.
Point Layer	A new layer is defined using the format CONPxx_xx. The xx_xx represents the elevation level of the contour string to two decimal places. The point nodes for the string are written to this layer.
Filter Points	Implements a filter to reduce the number of nodes created for the string.
Distance	Sets the distance for applying a filter.
Angle	Sets the angle for applying a filter.

4. If the *Add New Layer* dialog box appears, modify the properties if necessary, and click **OK**.
5. Press **Esc** or select **Cancel** from the right-click shortcut menu to end the command.

Create DTM Boundary

To create a DTM boundary inside or outside of a DTM, do the following:

1. Click **Surface** ► **DTM Tools** ► **Create DTM Boundary**.

The *Create DTM Boundary* dialog box appears (Figure 14-80).

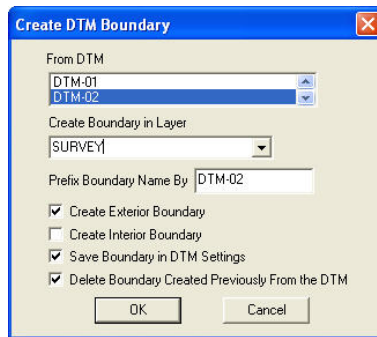


Figure 14-80. Create DTM Boundary Dialog Box

2. Set the necessary preferences, and then click **OK**.

Create Boundary

Use the Create Boundary command to create a boundary around an area that will be used for other computations, especially for volume computations where the boundary line defines the extent of the surface area used in the calculation. A boundary is a polygon that has a unique name.

A boundary may also be used for deleting triangles. A boundary can be created from existing points and lines, new points, or any combination of these. The boundary can be converted to a lot or pad at any time after it is created (Figure 14-81 on page 14-64).

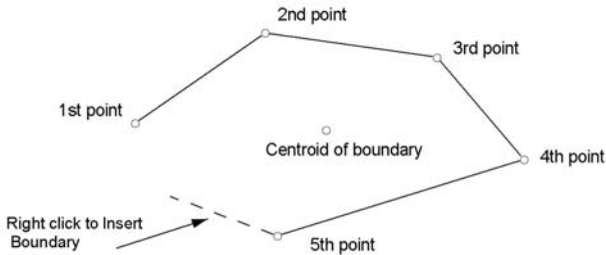


Figure 14-81. Creating a Boundary from Existing Points

To use the Create Boundary command, do the following:

1. Click **Surface ► Create Boundary**.
2. Select the first point or line as indicated in the prompt line on the Status Bar.
3. Select the next point, and continue this process until you have closed all of the points for the boundary. At any time, you can enter coordinates in the input bar on the lower left of the display area.
4. Right-click on the mouse and click **Insert Boundary** from the menu.

The *Edit Boundary* dialog box appears (Figure 14-82).

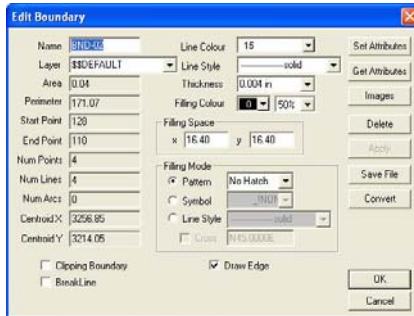


Figure 14-82. Edit Boundary Dialog Box

5. Modify the properties as necessary, and click **OK** to confirm boundary creation.

Table 14-14 describes the edit boundary fields.

Table 14-14. Edit Boundary Field Description



Field	Description
Name	Displays the default name for the boundary. The first boundary name is BND-01 with the suffix number increased for subsequent boundaries. You can edit this name.
Layer	Defines the layer on which the boundary is created.
Area	Displays the area enclosed within the boundary.
Perimeter	Displays the perimeter around the boundary
Points, Lines and Arcs	Definition of the boundary.
Centroid X and Y	Coordinate value of the centroid of the boundary.
Line Color	Defines the line color of the boundary edge.
Line Style	Defines the line style of the boundary edge.
Thickness	Defines the line thickness of the boundary edge.
Filling Color	The color of the fill for shading and patterning, and the shading percentage of the fill color.
Filling Space	The X and Y spacing for use with the Symbol fill or Line Style hatching.
Pattern	Choose Pattern to activate the drop-down menu, and then select a pattern or select No Hatch . The line color and fill color must be different for the patterns to display.
Symbol	Choose Symbol to activate the drop-down menu, and then select a symbol. The Point Symbols check box must be selected in the General Settings (Hot Button ) for the line style to display. The line angle is taken from the angle setting in the next field.
Line Style	Choose Line Style to activate the drop-down menu, and then select a line style. The line style entity must be selected in the General Settings (Hot Button ) for the line style to display. The line angle is taken from the angle setting in the next field.
Clipping Boundary	Check mark this box to use the boundary as a contour clipping boundary. The contours will not exist within the boundary.
Breakline	Creates a breakline from a boundary.

Table 14-14. Edit Boundary Field Description (Continued)

Field	Description
Draw Edge	Check mark this box to draw the edge of the boundary on the current layer.
Cancel	Press to quit the routine.
Set Attributes	Press to set the current project settings to the settings defined for the boundary.
Delete	Press to delete the highlighted boundary file.
Save File	Press to save the boundary for use as a Platform file.
Convert	Press to save the boundary as a platform in the current job.

Saving the Boundary as a Pad To save the boundary as a pad, do the following:

1. Click on the **Save File** button on the *Edit Boundary* dialog box. The *Save As* dialog box appears (Figure 14-83).

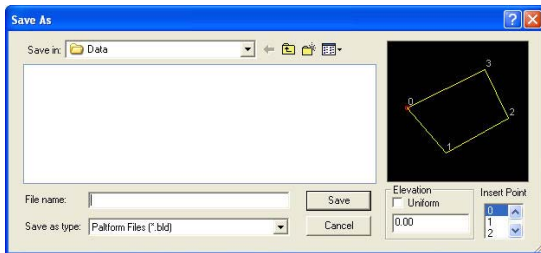


Figure 14-83. Save as a Pad File

2. Locate the folder in which you want to save the file.
3. Enter a name for the pad file.
4. Optionally, you can enter an elevation to apply to all points on the pad.
5. When you have finished, click **Save**.

A separate file is created in the folder you located. This file may be imported into another project. It has the extension .bld.

Convert to Polygon To convert the boundary to a polygon:

1. Click **Convert** on the *Edit Boundary* dialog box (Figure 14-82 on page 14-64).

The *Polygon Conversion* dialog box appears (Figure 14-84).

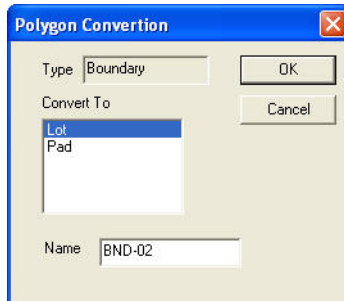


Figure 14-84. Polygon Conversion

2. Choose to convert to a **Lot** or a **Pad** and allocate a name.
3. Click **OK** to convert the polygon.

Create Enclosing Boundary

Use the Create Enclosing Boundary command to create a boundary which encloses a given set of points. The following methods are available to create an enclosed boundary: (Figure 14-85 and Figure 14-86 on page 14-68) and Figure 14-87 on page 14-69.

- Convex Boundary – creates a convex boundary around the extent of the selected points. The boundary will enclose all of the selected data.

NOTE: The boundary will be created on the active layer.

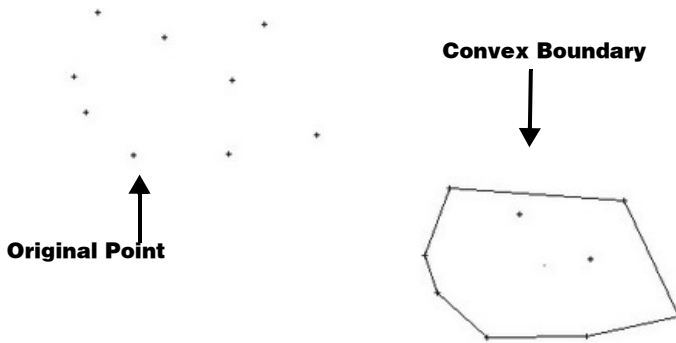


Figure 14-85. Convex Boundary Created to Enclose Extents of the Data

- Tight Boundary – creates a boundary passing through each of the outer points selected, thus forming a shrink wrap of the points (Figure 14-86).

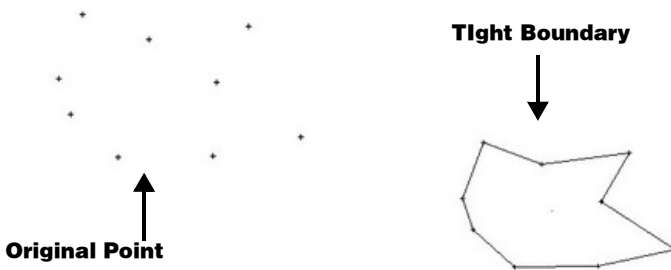


Figure 14-86. Tight Boundary Enclosed to Form a Shrink Wrap of Points

- Continuous Boundary – shrink wraps the selected points and entities to form a continuous boundary along the outer edges of the selection (Figure 14-87 on page 14-69).

NOTE: The boundary will be created on the active layer.

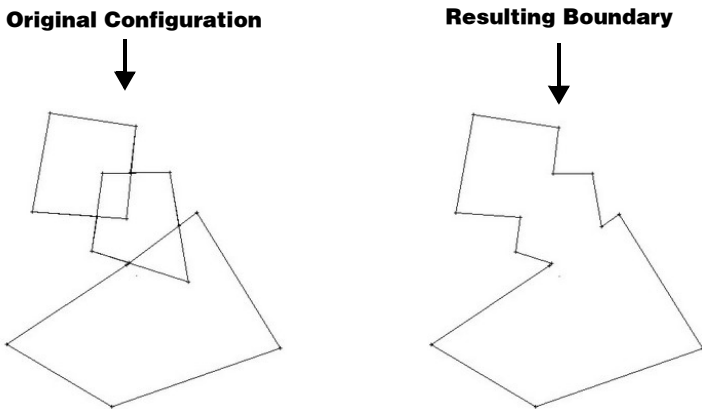


Figure 14-87. Continuous Boundary Shrink Wraps Selected Points/Entities

1. Click **Surface** ► **Create Enclosing Boundary**. The *Create Enclosed Polygon* dialog box appears (Figure 14-88 on page 14-70).
 - All – creates the selected boundary type around all points within the project.
 - Current Selection – creates only the boundary around the selected point.
 - Ignore points without elevation – check mark this point to exclude points without elevation in the project.
 - Method – enable one of three methods: *Convex*, *Tight*, or *Closed Boundary* (Figure 14-90 on page 14-72).
 - Convex Boundary: enable this method to create a convex boundary around the extent of the selected points. The boundary will be created on the active layer (Figure 14-91 on page 14-73).
 - Tight Boundary: enable this method to create a boundary that passes through each of the outer points selected, thus forming a shrink wrap of points. The boundary will be created on the active layer (Figure 14-91 on page 14-73).
 - Closed: enable this option to shrink wrap the selected points and entities to form a continuous boundary along the outer edges of the selection.

This option is also good to use when existing boundaries are present in the project. Using this routine detects the existing boundary and calculates the volume within its limits. The boundary will be created on the active layer.

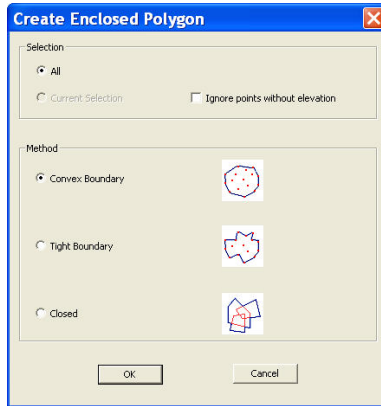


Figure 14-88. Create an Enclosed Polygon

2. Click **OK to save your changes and** to create the boundary.

NOTE: If no selection is made before the command is clicked, then **All** is the only available option.

See “Quick Volume” on page 14-82 for more information.

Create Boundary From Polygons

Use the Create Boundary from Polygons command to create a boundary from two or more existing polygons that lie against each other. There must be at least one common edge to any two of the selected polygons (Figure 14-89 on page 14-71).

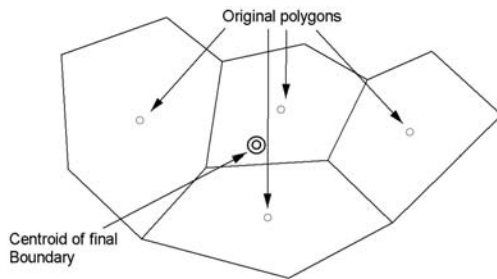


Figure 14-89. Four Polygons Converted to One Boundary

To create boundaries from polygons, do the following:

1. Click **Surface ► Create Boundary from Polygons**.
2. Select the mark at the centroid of an existing polygon.
3. Select the mark at the centroid of a polygon that has a common boundary with the first polygon.

A yellow boundary appears around both polygons.

4. If you want to include another polygon in the boundary, select the centroid of another polygon that has a common boundary with the currently displayed yellow boundary.
5. Continue until the selection is complete, and then select **Cancel** from the right-click shortcut menu or press **Esc**.

A boundary is saved from the extents of the connecting polygon.

Create Boundary List

Use the Create Boundary List command to create a list of several boundaries around data that will be used to compute volumes using the **Batch Volumes** command. This command defines a boundary for each closed area on the active layers displayed in the project.

Each boundary in the list is named by the layer on which its entities are found, with a unique number, as there may be more than one boundary on a layer.



Before you use this command, make sure the only active layers with closed areas are those required for the boundaries.

To use the Create Boundary List command, do the following:

1. Click **Surface ▶ Create Boundary List**.
The *Enter Boundary List Name* dialog box appears (Figure 14-90).

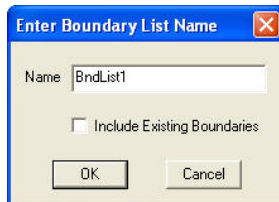


Figure 14-90. Enter Boundary List Name

2. Make any necessary modifications, and click **OK**. For a description of the fields, see Table 14-15 below.

Table 14-15. Enter Boundary List Name Field Descriptions

Field	Description
Name	The default boundary list name is displayed. The first boundary list name is BndList1 with the suffix number increasing for subsequent boundaries. You can change this name.
Include Existing Boundaries	Leave this check box empty to prevent duplication of existing boundaries. Select this check box if you want to include previously created boundaries in the list. The layers containing these boundaries must be active to ensure they are included. These may be boundaries that have already been created for another purpose.

A list of boundaries display (Figure 14-91).



Figure 14-91. Boundaries List

These listed boundaries have also been created as individual boundaries that you can use with other commands, like **Surface ▶ Compute Volume**.

Edit Boundary Lists

Use the Edit Boundary Lists command to delete a selected boundary list or remove individual boundaries from the list.

To delete a selected boundary list or remove individual boundaries from the list, do the following:

1. Click **Surface ▶ Edit Boundary Lists**.

The *Edit Boundary Lists* dialog box appears (Figure 14-92).

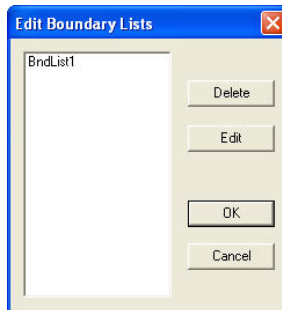


Figure 14-92. Edit Boundary Lists

2. To remove a boundary list from the project, select it and click **Delete**. A confirmation message appears.
3. Click **Yes**.

To delete an individual boundary from the list, do the following:

4. Select the list, and click **Edit**.
5. Select the boundary, and click **Delete**. No confirmation message appears.

Compute Volume

Use the Compute Volume command to calculate the volumes between two surface models within a defined boundary. The surface models may be in the same project or two projects. If the models are in two projects, both projects must be opened in MAGNET Office.

One of the surfaces may be a defined datum level to represent a surface as a horizontal plane. In this case, no DTM is required for this surface.

Several computational methods are supported and can be executed simultaneously, providing one report that contains the volumes calculated by all the selected methods. Each method provides cut, fill, and net volumes.

Computation Before computing volumes, take note of the following:

- DTMs must exist for both surfaces unless a datum height is being used for one surface.
- If the DTMs are in different projects, both projects must be opened in MAGNET Office and the design surface project should be the current project.
- A boundary must be defined for the computation. If two projects are being used, the boundary should be in the current project.
- DTMs for both surfaces should fully cover the boundary area. If the triangular mesh does not cover the full extent of the boundary, the results will be unreliable. You should check the boundary areas specified in the output report. The areas for the natural and design surfaces should match the boundary area.
- For accurate results, consider how well the mesh fits the actual terrain. If it is a good fit, then good results will be obtained.

- Make sure that all breaklines have been defined and inserted into the DTM; otherwise, the volumes computed may be inaccurate. Make sure there are no crossing breaklines.

Prism Method This method calculates exact volumes, as it uses of the DTM triangles to calculate the volumes (Figure 14-93). The mesh has been computed from the exact data, so no interpolation is used in the computation.

A brief explanation of how this method calculates volumes is as follows.

The triangular mesh for one project is subdivided until each triangle for that project fits entirely inside a triangle for the other project and is all cut or all fill. For each triangle, a mean height and area are computed. Because the triangles in both projects are coincident, individual cut and fill volumes can be built up over the entire model. Figure 14-93 illustrates the subdivision of triangles. The original triangulation is not altered by this subdivision; the subdivided triangles are only used in the computation and are then discarded.

$$Volume (Tri)_x = Area (Tri)_x \times (Mean Hgt Tri2 - Mean Hgt Tri1)_x$$

Figure 14-93. Prism Method Formula for Computing Volumes

Individual cut and fill volumes are then tabulated to produce total cut and total fill for the entire area within the boundary (Figure 14-94).

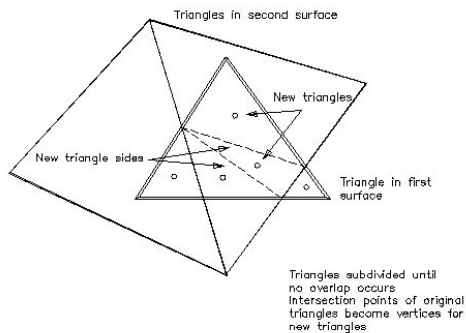


Figure 14-94. Prism Method for Computing Volumes

The prism method is the most accurate method of determining volumes in MAGNET Office. The surveyed points are used for the

triangular model, so there is no interpolation of data giving the closest result for the project data.

Slices Method This method computes volumes between specified contour intervals using the prismoidal method (Figure 14-96).

The surface area of the two models is in the form of 3-dimensional triangles. The slices method subdivides this existing triangular mesh until each triangle for the project fits entirely inside a triangle for the other project, is all cut or all fill, and is within the defined boundary. The volume for each subdivided triangle is bounded by up to seven planes; the three vertical sides, the natural surface, the design surface, and the two contour intervals defined by the slice interval, as shown Figure 14-96.

MAGNET Office uses these subtriangle prisms to calculate volumes for each subtriangle prism and for each relevant slice. The results for each slice are accumulated.

The formula used to calculate each individual slice in the triangular prism is shown in Figure 14-95.

$$\text{Volume (Tri)} = \text{Area (Tri)} \times \text{average delta } z$$

Figure 14-95. Slices Method Formula for Computing Volumes

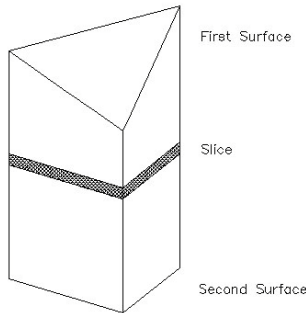


Figure 14-96. Slices Method for Computing Volumes

The final report includes volumes between contours at the nominated interval and is suited to stockpiles, excavations, storm water ponds, dams, and reservoir computations. In the case of an excavation, the result will include cut and fill volumes, but the required answer will

be either the total cut or total fill, depending on which surface is entered as the design surface.

Grid Method In this method, a rectangular grid is overlaid onto the data (Figure 14-98). At each grid node (intersection) a height is computed for each surface by interpolation from the triangular mesh. The grid spacing in both X and Y directions may be chosen. The accuracy of the answer will increase with a decrease in the grid spacing. The grid interval chosen should be relevant to the terrain involved. For example, for an open pit, it is recommended that the grid interval should be 1/5th the size of the smallest triangle.

This method can only give approximate results because the computation is made from interpolated points. The grid square is subdivided into two triangles for each surface.

The delta z's for each vertex of the triangles are calculated and the formula in Figure 14-97 is used to calculate the volume.

$$Volume (Tri) = Area (Tri) \times average \ delta \ z$$

Figure 14-97. Grid Method Formula for Computing Volumes

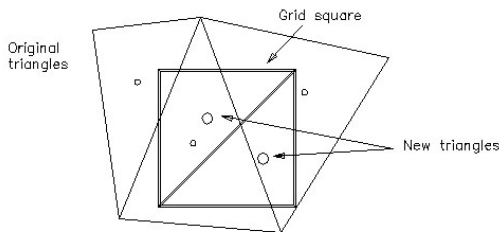


Figure 14-98. Grid Method for Computing Volumes

If the grid triangles are not all cut or all fill, then these triangles are further subdivided and the interpolation of the heights for these triangles is extracted from the grid levels, not the triangular mesh. The same formula is applied to these triangular prisms.

Subdivision of triangles also occurs if one of the grid points is outside the boundary. Triangles are formed to scissor against the boundary as shown below. If the height of the point outside the boundary is known (interpolated from the triangular mesh), then heights along the

boundary will be a linear interpolation from the existing grid points. If the point outside the boundary is unknown, then heights along the boundary will be interpolated from the triangular mesh (Figure 14-99).

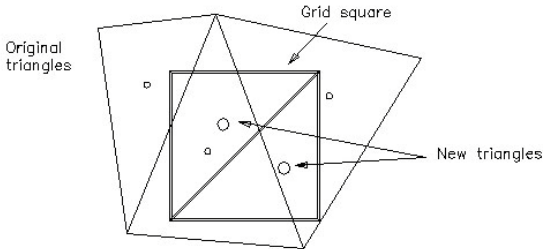


Figure 14-99. Grid Points Outside the Boundary

To use the Compute Volume command, do the following:

1. Click **Surface ▶ Compute Volume**.

The *Standard Volumes* dialog box appears (Figure 14-100).

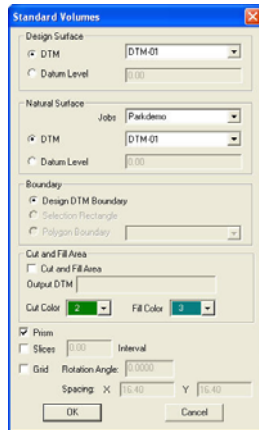


Figure 14-100. Standard Volumes Dialog Box

2. Modify the properties as needed, and then click **OK**. For a description of the standard volumes fields, see Table 14-16 on page 14-79.

A report listing the computed volumes displays. You can read, print, or save the report to a report (*.rep*) file as a record of the computation.

Table 14-16. Standard Volumes Field Descriptions

Field Name	Field Description
DTM (Design Surface)	Select this option to use a STM model as the design surface of the computation. Select the required DTM from the drop-down menu of the DTMs of the current project.
Datum Level (Design Surface)	Select this option to use a known datum level for the design surface of the computation. Enter the datum level in the adjacent field.
Job (Natural Surface)	The current project is displayed. If the natural surface is in another project, select the project from the drop-down menu. Only the projects opened in MAGNET Office are on the list.
DTM (Natural Surface)	Select this option to use a DTM model as the natural surface of the computation. Select the required DTM from the drop-down list of the DTMs available for the project defined in the Jobs field.
Datum Level (Natural Surface)	Select this option to use a known datum level for the natural surface of the computation. Enter the datum level in the adjacent field.
Boundary	Select Polygon Boundary to use a previously created boundary.
Cut and Fill Area	Select this check box, and then choose cut and fill colors to show the location and the amount of cut and fill in the output DTM. Enter a name for the output DTM.
Prism	Computes volumes using the prismatic method. See also “Prism Method” on page 14-75.
Slices	Computes volumes using the slice method. Type in the contour interval used for the slices. See also “Slices Method” on page 14-76.
Grid	Computes volumes using the grid method. Complete the following fields: Rotation Angle: Modify this value to change the rotation of the grid over the surface model. Spacing X and Y: Enter the spacing between grid nodes in the X and Y directions.

Batch Volumes

Use the Batch Volumes command to monitor progressive volumes on site or calculate volumes for numerous stockpiles within one project. The Batch Volumes command calculates volumes, using the prism method, between two surfaces within the same project or between two projects for each boundary in a defined boundary list. See “Create Boundary List” on page 14-71.

Before using batch volumes, take note of the following:

- DTMs must exist for both surfaces.
- If the DTMs are in different projects, both projects must be opened in MAGNET Office and the design surface project should be the current project.
- A boundary must be defined for the computation. If two projects are being used, the boundary should be in the current project.
- DTMs for both surfaces should fully cover the boundary area. If the triangular mesh does not cover the full extent of the boundary, the results will be unreliable.
- Make sure that all breaklines have been defined and inserted into the DTM; otherwise, the volumes computed may be inaccurate. Make sure there are no crossing breaklines.

To calculate volumes, do the following:

1. Click **Surface** ► **Batch Volumes**.

The *Batch Volume Calculation* data entry dialog box appears (Figure 14-101).

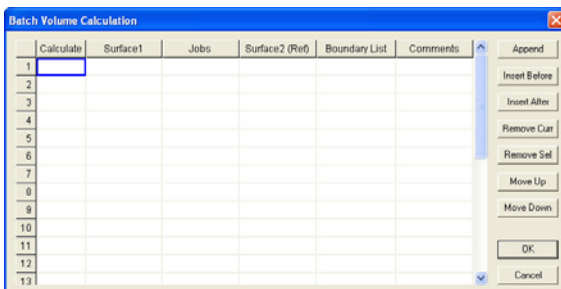


Figure 14-101. Batch Volume Calculation

- Fill in one line of data for each computation that uses one boundary list, and then click **OK**. For field descriptions, see Table 14-17 on page 14-81.

A report displays, listing the volume computation for each boundary in the boundary list. You can read, print, or save the report to a report (.rep) file as a record of the computation.



Always check that the sum of the cut, fill, and matched areas adds up to the boundary area. If they do not, then make sure the triangles on the surfaces reach the boundary edges.

Table 14-17. Batch Volume Calculation Field Descriptions

Field	Description
Calculate	Includes this line in the computation and report
Surface1	Select the DTM from the drop-down menu. This is a STM in the current design project.
Jobs	If the natural surface is in another project, select the project from the drop-down menu. Only the projects opened in MAGNET Office will be on the list.
Surface2 (Ref)	Select the DTM from the drop-down menu. This is the reference DTM and usually the natural surface. It may be another project if a project is selected in the Jobs column.
Boundary List	Select the list containing the boundaries for this computation from the drop-down menu. The boundary list must be in the current project.
Comments	Enter an optional comment for display on the report.
Buttons	
Append	Adds a new line to the end of the list of items on the Batch Volume Computation dialog box.
Insert Before	Highlight a line, and click Insert Before to create a new line for data before the highlighted line.
Insert After	Highlight a line, and click Insert After to create a new line for data after the highlighted line.
Remove Current	Removes the line where the pointer is currently positioned.
Remove Sel	Removes the line which is currently highlighted.
Move Up	Moves the selected line of data above the previous line.

Table 14-17. Batch Volume Calculation Field Descriptions (Continued)

Field	Description
Move Down	Moves the selected line of data below the next line.

Quick Volume

The quick volume routine automatically creates an existing DTM and a design DTM, then calculates a volume between the two surfaces. You must have data that defines, for example, both the toe of a stockpile and the stock pile itself.

To automatically create an existing DTM and a Design DTM, click **Surface ▶ Quick Volume**. The *Quick Volume* dialog box displays (Figure 14-102).

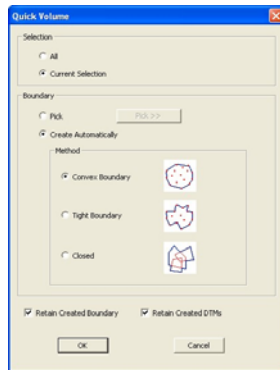


Figure 14-102. Quick Volume Dialog Box

There are three elements to define for this routine: *Selection*, *Boundary*, and *Retention*.

- Selection element:
 - All: enable this option to use all visible entities in the Survey view to create a boundary that encompasses all of the data.

The type of boundary selected will determine the configuration of the created boundary. For example, Figure 14-103 on page 14-83 shows a Convex Boundary and Tight Boundary that was created around all visible data.

NOTE: This option uses only preselected data to form a boundary.

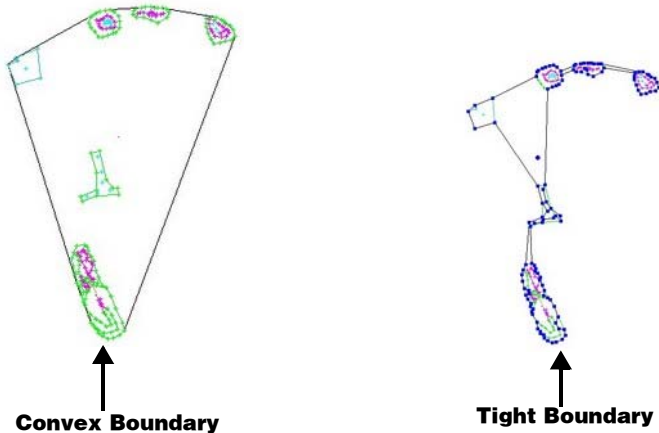


Figure 14-103. Quick Volume – Create a Selection Element

- Boundary element:
 - Pick: enable this option to geographically select an existing boundary from the Survey view.
 - Create Automatically: enable this option to have the program automatically create the boundary, based upon the boundary creation method selected.
 - Method:
 - Convex Boundary: this method will create a convex boundary around the extent of the selected points (Figure 14-104 on page 14-84). The boundary will enclose all of the selected. The boundary will be created on the active layer.
 - Tight Boundary: this method will create a boundary passing through each of the outer points selected, thus forming a shrink-wrap of the points (Figure 14-104 on page 14-84). The boundary will be created on the active layer.

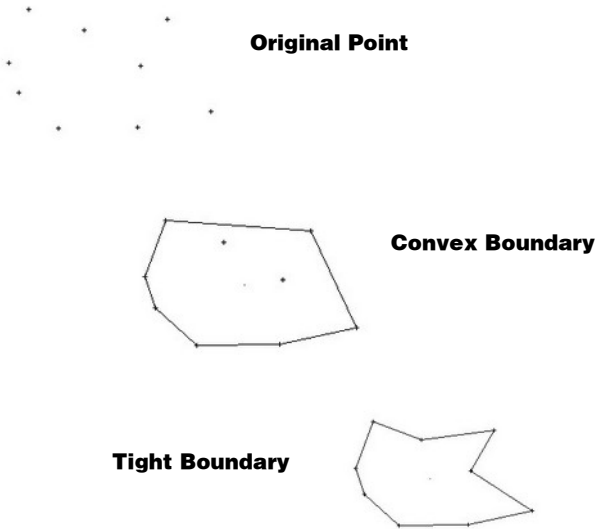


Figure 14-104. Example – Convex and Tight Boundary

- Closed: this method will shrink-wrap the selected points and entities to form a continuous boundary along the outer edges of the selection. The boundary will be created on the active layer.

This method is also a good option to use when existing boundaries are present in the project. The routine will detect the existing boundary, then calculate the volume within its limits.

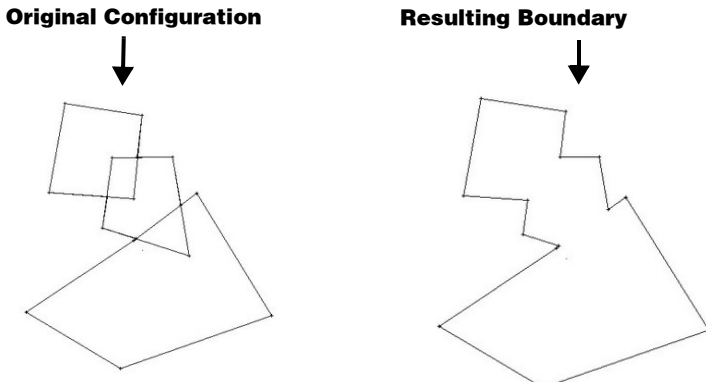


Figure 14-105. Example – Closed Boundary

- Retain Created Boundaries – enable this option to retain the boundary and place it on the active layer specified in the Survey view.
- Retain Created DTMs – enable this option to save the DTMs that are created during the volume calculations. The DTM is named automatically using the following identifiers:
QuickVolume_Base_0 and *QuickVolume_Pile_0*.

NOTE: The number identifier at the end of the DTM name will increase as additional DTMs are created and retained using the *Quick Volume* routine. For example, the next quick volume calculation that retains the DTMs will result in the DTM identifiers of *QuickVolume_Base_1* and *QuickVolume_Pile_1*.

Survey View: Tools Menu

This section describes the different Survey View menu commands on the Tools menu located on the main menu bar (Figure 15-1).

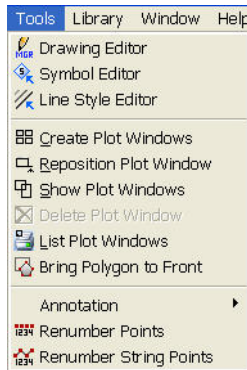



Figure 15-1. Survey View – Tools Menu Options

In This Section:

- “Using the Drawing Editor” on page 15-2
- “Symbol Editor” on page 15-2
- “Line Style Editor” on page 15-4
- “Create Plot Windows” on page 15-6
- “Reposition Plot Window” on page 15-8
- “Show Plot Windows” on page 15-9
- “List Plot Windows” on page 15-9
- “Bring Polygon to Front” on page 15-9
- “Modify an Annotation” on page 15-10

- “Renumber Points” on page 15-16
- “Renumber String Points” on page 15-19

Using the Drawing Editor

In the drawing editor, you can add information such as lines, arrows, and text, and you can edit the title area as required, but the survey information cannot be altered (linked drawing). Make sure to check the **Hot Button**  dialog box in the drawing editor to set items to display. There is also a Locked flag in the **Hot Button** dialog box controlling the title block and the title area. Clear this check box if you want to edit the title area.

See also “Roads View Editor” on page 18-1.

Symbol Editor

Use the Symbol Editor command to create and modify a symbol in the Symbol Editor. In the Symbol Editor, the symbols are displayed at a magnified size. The symbol is displayed with a red point at its center and a red dotted line displaying the extent of the current clipping.

To create and modify a symbol, do the following:

1. Click **Tools** ▶ **Symbol Editor**.

The **Symbol Manager** dialog box appears, displaying the symbols used in the current project (Figure 15-2 on page 15-3).

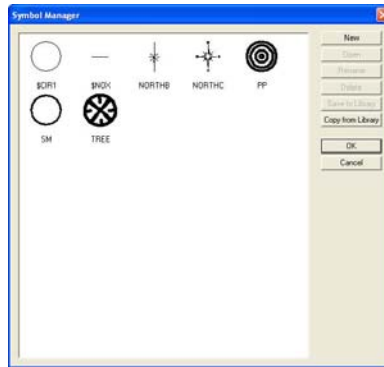



Figure 15-2. Symbol Editor

2. To add a symbol to the *Symbol Manager* dialog box from the library, click **Copy from Library**, select a symbol, and then click **OK**.

The symbol appears in the *Symbol Manager* and can be edited without altering the symbol in the library.

3. To edit a symbol, double-click on the symbol or select it and click **Open**. The symbol appears in the **Symbol Editor** (Figure 15-3 on page 15-4) enclosed in a red dotted square. Modify the symbol using the tools on the toolbar, and then complete the symbol definition by clicking the **Hot Button** . Make the appropriate selections, and then click **OK**.

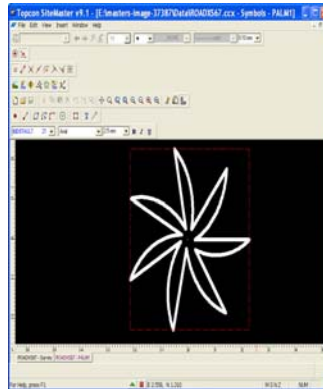


Figure 15-3. Symbol Editor

4. To rename, delete, or save a symbol to the library, select it and click the appropriate button.
5. When you have finished, click **OK**.

Line Style Editor

Use the Line Style Editor command to create and modify line styles in the Line Style Editor. In the Line Style Editor, the lines display at a magnified size. The line displays with only one repeat of the style on the screen and a red dotted line displays the extent of the repeat. This may include a space between the end of one repeat and the start of the next.

To create and modify line styles, do the following:

1. Click **Tools ▶ Line Style Editor**.

The **Line Style Manager** appears (Figure 15-4 on page 15-5), displaying the symbols used in the current project.

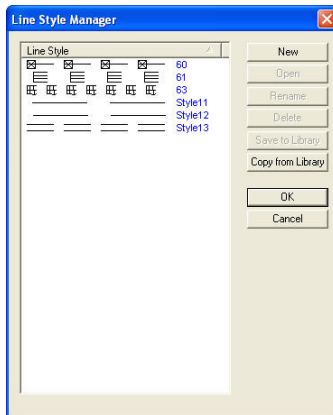



Figure 15-4. Line Style Dialog Box

2. To add a line to the **Line Style Manager** from the library, click **Copy from Library**, select a line, and then click **OK**. The line appears in the **Line Style Manager** (Figure 15-5) and can be edited without altering the line in the library.
3. To edit a line, double-click on the line or select it and click **Open**. The line appears in the **Line Style Editor** enclosed in a red dotted line. Modify the line using the tools on the toolbar and then complete the line definition by clicking the **Hot Button** . Make the appropriate selections, and then click **OK**.

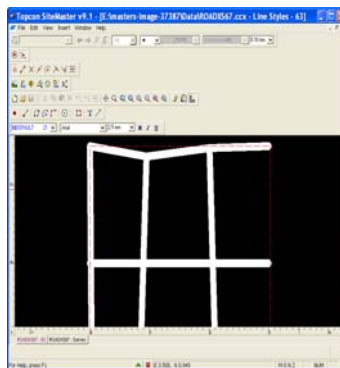


Figure 15-5. Line Style Editor

4. To rename, delete, or save a line to the library, select it and click the appropriate button.

5. When you have finished, click **OK**.

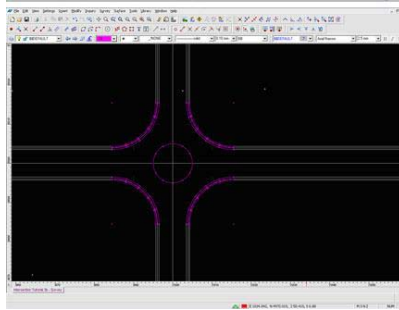


Figure 15-6. Overlapping Strings Removed

Create Plot Windows

Use the Create Plot command to create a linked drawing that is displayed in the Drawing View. The drawing is defined with a nominated plot scale, title block, and orientation. The drawing displays all of the data currently on active layers in the Survey View, complete with any annotation, triangles, and contours activated in the **General Settings** (Hot Button) of the project.

You can turn layers on or off in the drawing. Any layer added to the project after the drawing was created may be turned on in the drawing. Any annotation that was not available at the time the drawing was created, may be activated using the commands in the **Drawing View**.

If entities are added, removed, or edited on the layers that were active when the drawing was first created, they will be updated automatically in the linked drawing.

Several plot windows can be defined in one project, creating a linked drawing for each created plot window. The drawings may be saved with unique names to indicate the drawing content.

To create a linked line, do the following:

1. Click **Tools** ► **Create Plot Windows**.

The *Open* dialog box appears (Figure 15-7 on page 15-7).

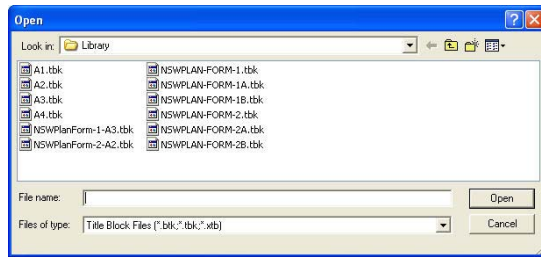


Figure 15-7. Open Window

2. Locate the required title block for this drawing, and click **Open**. The lower-left corner of the title block displays in the middle of the screen (Figure 15-8).
3. Use the zoom options and mouse to position the window over the data.

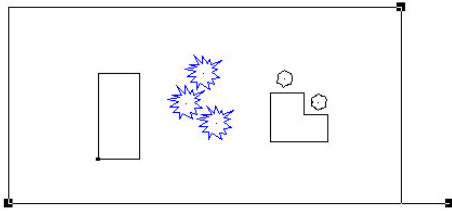


Figure 15-8. Drawing Area of Title Block Dragged Around Data for Drawing


4. Modify the settings and position the title block around the data on the screen.
 - The mouse can drag the plot window and snap the lower-left handle to an existing point.
 - The mouse can rotate the plot window using the right-lower handle, the arrow handle, or it can be rotated parallel to a line.
 - The plot window can be resized to find an approximate scale, using the upper-right drag handle.
5. When the plot window is correctly defined, click **OK** on the input bar (Figure 15-9).



Figure 15-9. Title Block Input Bar

The new drawing is created and opened in the Drawing View. The Survey View window remains open but the Drawing View is the active window.

Additional Information

- A series of plot windows may be defined in a project. These windows may overlap each other or be created at different scales. Each drawing is initially created as Drawing X, where X is the next available number. The drawing may be allocated a more suitable name in the Drawing View.
- Once a drawing has been created, it may be reopened from the list of drawings in the **Drawings View** .

Reposition Plot Window

Use the Reposition Plot command to change the position, scale, and/or title block that was used to define an existing linked drawing.

To change the position, scale, and/or title block, do the following:

1. Click **Tools ▶ Reposition Plot Window**.

The *Select Drawing* dialog box appears (Figure 15-10).



Figure 15-10. Select Drawing

The drop-down menu contains all plot windows created in the Survey View and held as linked drawings in the Drawing View.

2. Select a drawing to reposition from the drop-down menu, and then click **OK**.

The settings displayed in the input bar are the original settings for the plot window. The plot window is displayed over the project data in the Survey View.

3. Use the input bar to make modifications or use the mouse to reposition the window, and then click **OK** on the input bar to save the changes.

Show Plot Windows

Use the Show Plot command to turn on and off the position of the currently defined plot windows in the project.

To use the Show Plot Windows, do the following:

1. Click **Tools ▶ Show Plot Windows** or press the **F10** key.
The plot windows display in yellow and a check mark appears next to the **Show Plot Windows** command on the **Tools** menu. Repeat this routine to turn the command off.
2. Double-click inside a plot window to open the drawing as the active window.

List Plot Windows

To produce a report that lists all of the plot windows in a project, click **Tools ▶ List Plot Window**. The List Plot Windows report displays, listing the project name, drawing name, TB name, scale, rotation, and extents.

Bring Polygon to Front

Use the Bring Polygon to Front command to bring a nominated polygon to the front of other polygons, so the detail of the selected polygon overwrites the detail of other polygons defined in the same area (Figure 15-11 on page 15-10). Polygons are usually displayed in the order in which they were created. This routine allows you to change that order.

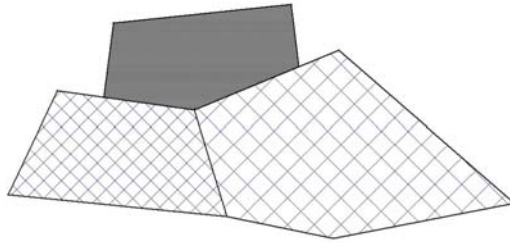


Figure 15-11. Cross Hatched Polygons Brought in Front of Shaded Polygon

To bring a nominated polygon to the front of other polygons, do the following:

1. Click **Tools ▶ Bring Polygon to Front**.
2. Select a polygon. MAGNET Office will display this polygon on top of other polygons.
3. Continue to reorder polygons as required, and then press **Esc** to quit the routine.

Modify an Annotation

The commands on the *Annotation* submenu contain options for modifying the annotation currently defined for the various entities in the project.

The selected annotation may be updated to use a different text style, relative position, or rounding computation. The changes made to the annotation cannot be reversed with the **Undo** command, but can be changed using the Recreate command.

The annotation settings are allocated to an entity when its annotation check box is selected. Changes may be needed to update the annotation, especially if the proposed scale is modified.

In This Section:

- “Redefine an Existing Annotation” on page 15-11
- “Delete an Existing Annotation” on page 15-12
- “Annotation Table Settings” on page 15-13

Redefine an Existing Annotation Use the Recreate command to redefine the existing annotation. It updates the annotation to apply the changes made to the current styles set in the Annotation Table Settings.

To redefine the existing annotation, do the following:

1. Click **Tools ▶ Annotation ▶ Create**.

The *Create Annotation* dialog box appears (Figure 15-12).

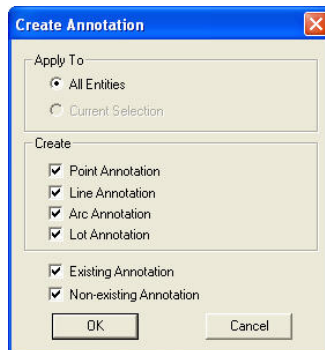


Figure 15-12. Create Annotation

2. Modify the properties as needed, and then click **OK** to recreate the annotation. For a description of the fields, see Table 15-1 below.

Table 15-1. Create Annotation Field Descriptions

Field	Description
All Entities	Select this option to apply the changes to all of the entities on the current layers.
Current Selection	If data is selected before using the Recreate command, then this option will be automatically chosen.
Point Annotation	Applies changes to all points in the selection. The annotation settings currently defined in the Point Annotation Settings will be used to recreate only existing annotations active for the selected points. If the Non-existing Annotation check box is also selected, the annotation is activated for all the points in the selection.


Table 15-1. Create Annotation Field Descriptions (Continued)

Field	Description
Line Annotation	<p>Applies changes to all lines in the selection. The annotation settings currently defined in the Line Annotation settings (with Bearing and Distance Tables) will be used to recreate only existing annotations active for the selected lines.</p> <p>If the Non-existing Annotation check box is also selected, the annotation is activated for all the lines in the selection.</p>
Arc Annotation	<p>Applies changes to all arcs in the selection. The annotation settings currently defined in the Arc Annotation settings (with Bearing and Distance Tables) will be used to recreate only existing annotations active for the selected arcs.</p> <p>If the Non-existing Annotation check box is also selected, the annotation is activated for all the arcs in the selection.</p>
Lot Annotation	<p>Applies changes to all arcs in the selection. The annotation settings currently defined in the Lot Annotation settings (with Bearing and Distance Tables) will be used to recreate only existing annotations active for the selected lots.</p> <p>If the Non-existing Annotation check box is also selected, the annotation is activated for all the lots in the selection.</p>
Existing Annotation	<p>Applies the changes to the entities selected in the Create check boxes, which have existing annotations defined. It will have no effect on entities with no active annotation flags.</p> <p>Use this option to update the current annotation to new settings.</p>
Non-existing Annotation	<p>Applies the changes to the entities selected in the Create check boxes. Use this option with caution, because the Undo command does not apply to the setting of annotation flags on entities.</p>

Delete an Existing Annotation Use the Delete command to delete the existing annotation for all entities in the current selection. This routine removes the check marks from the boxes next to the **Annotation** fields for these entities.

To delete the existing annotation for all entities, do the following:

1. Select the entity whose annotation is to be removed.
2. Click **Tools** ▶ **Annotation** ▶ **Delete**.

3. Select the **Regen**  icon or **View ▶ Regen** to clear any remaining annotation from the screen display.



Annotation Table Settings

Use the Annotation Table Settings command to control the format of line and arc annotation tables. Any line can have its annotation plotted in a table instead of along or adjacent to the line. This reduces the clutter on the drawing. It is most useful for short lines where the annotation does not fit neatly into the line length.

Any arc can have its annotation plotted in a table instead of adjacent to the line. This is often advisable if a number of the arc elements are to be annotated, as there is rarely room to include all of the elements adjacent to the arc on the drawing.

Only lines and arcs with their Annotation Table check box selected will be included in the table. This option is used to specify the format, layout, and position of the line and/or arc annotation tables in the project.

To control the format of line and arc annotation tables, do the following:

1. Make sure to check mark the **Line Annotation Table** and **Arc Annotation Table** boxes on the *Display* tab of the **General Settings** (Hot Button ) dialog box to display the tables. To access the dialog box, click the **Hot Button**  icon on the bottom of the window.
2. Click **Tools ▶ Annotation ▶ Annotation Table Settings**.

The *Annotation Table Settings* dialog box appears (Figure 15-13 on page 15-14).

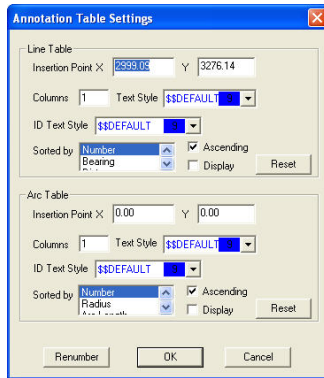


Figure 15-13. Annotation Table Settings Dialog Box

3. Modify the properties as necessary, and then click **OK** to display the tables. For field descriptions, see Table 15-2 below.
See also “Lines” on page 15-15 and “Arcs” on page 15-16.

Table 15-2. Annotation Table Settings Field Descriptions

Field	Description
Line Table	
Insertion Point X, Y	Defines the X, Y coordinates. This can be modified to ensure the table is included in a plot window at a later time.
Columns	Select the number of columns in the table. Enter 1 to list the lines and their bearings and distances in one columns. Enter 2 to list the lines and their bearings and distances in two columns, and so on.
Text Style	Select the text style from the drop-down menu for the data in the table.
ID Text Style	Select the text style from the drop-down menu for the ID number that will annotate the line in the project. Each line will have its ID number written beside the line. This number is written in a circle.
Sorted By	The table will be sorted in the order of the item selected from the drop-down menu. Number: The table is sorted by the line ID number. Bearing: The table is sorted by the bearing of the line. Distance: The table is sorted by the distance of the line.

Table 15-2. Annotation Table Settings Field Descriptions (Continued)

Field	Description
Ascending	Lists the table in ascending order of the property selected in the Sorted By list.
Display	Displays the line table in the Survey View at the coordinate specified.
Arc Table	
Insertion Point X, Y	Defines the X, Y coordinates. This can be modified to ensure the table is included in a plot window at a later time.
Columns	Select the number of columns in the table. Enter 1 to list the arcs and their parameters in one columns. Enter 2 to list the arcs and their parameters in two columns, and so on.
Text Style	Select the text style from the drop-down menu for the data in the table.
ID Text Style	Select the text style from the drop-down menu for the ID number that will annotate the line in the project. Each line will have its ID number written beside the line. This number is written in a circle.
Sorted By	The table will be sorted in the order of the item selected from the drop-down menu. Number: The table is sorted by the arc ID number. Radius: The table is sorted by the arc radius. Arc Length: The table is sorted by the arc length. Angle: The table is sorted by the arc angle. Chord Bearing: The table is sorted by the chord bearing of the arc. Chord Distance: The table is sorted by the chord distance of the arc.
Ascending	Lists the table in ascending order of the property selected in the Sorted By list.
Display	Displays the line table in the Survey View at the coordinate specified.
ReNUMBER	Renumbers all of the lines and arcs.

Lines To appear in a short line table, the line must have the **In Line Annotation Table** box checked in the *Edit Line* dialog box. To access the *Edit Line* dialog box, select a line and choose **Properties** from the right-click menu.

If the **Show Short Line Annotation** box is not checked in the Line Annotation Settings (**Settings ▶ Annotation Settings ▶ Line Annotation Settings tab**), then it will automatically be selected as short lines. These lines will have the **In Line Annotation Table** box checked to place them into the Line Table.

Arcs To appear in an arc table, the arc will have the **In Arc Annotation Table** box checked in the *Edit Arc* dialog box.

The Arc Table reads the current Arc Annotation Settings style (**Settings ▶ Annotation Settings ▶ Arc Annotation Settings tab**) and will only list the arc parameters that are selected for the annotation in this style.

If the parameters have been modified in the style but full detail is not shown in the table, then turn off the annotation flags for these arcs and turn them on again.

Renumber Points

Use the Renumber Points command to renumber all of the points in a project or all of the points in a selection. Use this routine to rename alphanumeric points for upload into a data logger or to renumber a selection of points to another point number range.

If no points are selected, then all of the points in the project, including points on inactive layers, will be renamed or renumbered.

If only certain entities are to be renumbered, then select the entities before using this command.



Be aware of the results of renumbering points, especially if the data is interfaced with other software or with imported or downloaded datasets.

To renumber all of the points in a project, do the following:

1. Click **Tools ▶ Renumber Points**.

The *Renumber Points* dialog box appears (Figure 15-14).

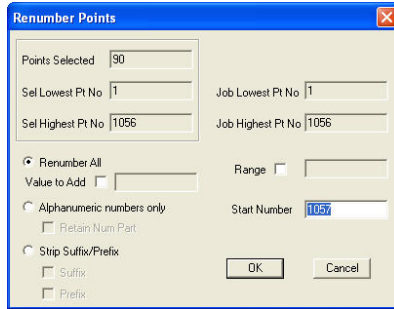


Figure 15-14. Renumber Points

2. Modify the properties as needed. For field descriptions, see Table 15-3 below.
3. When you have finished, click **OK**.

A report displays, listing the original point numbers in the selection and the new point numbers allocated to those points. You can read the report on the screen, print it, or save it to a report (.rep) file.



If the numbers are not suitable or there are error messages, click **Edit ▶ Undo** to return the numbers to their previous settings.

See also “One Point” on page 15-18.

Table 15-3. Renumber Points

Field	Description
Points Selected	Displays the number of points selected for renumbering.
Sel Lowest Pt No	Displays the lowest point number in the selection.
Sel Highest Pt No	Displays the highest point number in the selection.
Job Lowest Pt No	Displays the lowest point number in the project.
Job Highest Pt No	Displays the highest point number in the project.
Renumber All	The default setting. Renumbers all points in the selection with the lowest point starting at the number in the Start Number field.

Table 15-3. Renumber Points (Continued)

Field	Description
Value to Add	<p>Represents the value added to the existing numbers. Select the check box, and enter a number in the adjacent field. All the points in the selection will have the offset added to their existing point numbers. The offset = 1000, so 1000 is added to all point numbers. For example: point 25 becomes 1025 and point 978 becomes 1978.</p> <p>An alphanumeric value may be used as the offset. It is added as a prefix to the existing number. The offset = AB, so that point 25 becomes AB25 and point 978 becomes AB978.</p>
Alphanumeric Numbers Only	<p>Applies the renumbering only to alphanumeric point numbers in the selection. These numbers will be renumbered with the lowest point in the selection starting at the number in the Start Number field.</p>
Retain Num Part	<p>Removes the letters from the number, leaving on numbers for each point. If there is already a point with only numbers, a new point number is allocated. NZ25 becomes 25 as long as no point number 25 exists.</p>
Strip Suffix/Prefix	<p>Removes the suffix and/or prefix, depending on the option you select, from a point number.</p>
Range	<p>Renumbers only the point numbers in the specified range. Select the check box, and enter a range of point numbers in the adjacent field. The range is an additional filter and is useful if you did not select points before using this command.</p> <p>The Range format is: 1-100 (renumber all points from 1 to 100) or 151-200, 300-350, 360-370 (renumber all points 151 to 200, 300 to 350, and 360 to 370). Other points will not be renumbered.</p>
Start Number	<p>The next available number after the project's highest number is displayed. This is used for renumbering points. This number is usually the Next Point Number set in the General Settings (Hot Button). This number may be changed to another available number.</p>

One Point To renumber one point in a project:

1. Select it and click **Tools ▶ Renumber Points** in the Survey View. The *Renumber Points* dialog box appears (Figure 15-15).



Figure 15-15. Renumber Points

2. Modify the properties as necessary, and then click **OK** to change the number of the selected point.

Renumber String Points

Use the Renumber String Points command to renumber all of the points in a string line of connected points and lines or in an existing string entity, so the nodes of the string are numbered in sequential order along the length of the string.

To renumber all of the points in a string line of connected points, do the following:

1. Click **Tools ▶ Renumber String Points**.
2. Select a string or points to define as string as stated in the prompt line.
3. Continue selecting points of the string and when you have finished, select **Insert String** from the right-click shortcut menu.

The *Renumber Points* dialog box appears (Figure 15-16 on page 15-20).

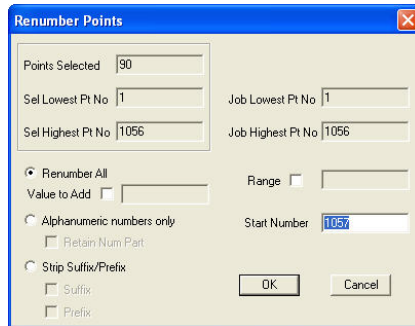


Figure 15-16. Renumber Points

4. Make sure **Renumber All** is selected and enter a value in the **Start Number** field.
5. Click **OK**.

A report displays, listing the original point numbers in order along the string and the new point numbers allocated to those points. You can read the report on the screen, print it, or save it to a report (.rep) file.



If the numbers are not suitable or there are error messages, click **Edit ▶ Undo** to return the numbers to their previous settings.

Survey View: Library Menu

This section describes the different Survey View menu commands on the Library menu located on the main menu bar (Figure 16-1).

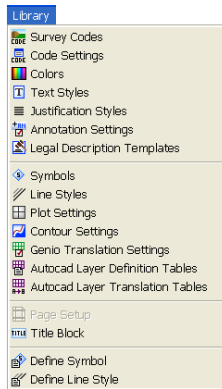


Figure 16-1. Survey View Menu – Library Menu

In This Section:

- “Set Up Survey Codes” on page 16-2
- “Code Settings” on page 16-12
- “Create Custom Colors” on page 16-17
- “Text Styles” on page 16-18
- “Justification Styles” on page 16-21
- “Annotation Styles” on page 16-24
- “Legal Description Templates” on page 16-24
- “Symbols” on page 16-29
- “Line Styles” on page 16-34

- “Plot Settings” on page 16-39
- “Contour Settings” on page 16-46
- “Genio Translation Settings” on page 16-48
- “Autocad Layer Definition Tables” on page 16-50
- “Autocad Layer Translation Tables” on page 16-52
- “Page Setup” on page 16-55
- “Title Block” on page 16-56
- “Symbol” on page 16-59
- “Define Line Style” on page 16-61

Set Up Survey Codes

Use the Survey Codes command to set up the survey feature codes used in the field to label the observations made that are recorded in a data logger, GPS recorder, other instrument, or field book. Each feature code has attributes defined for layer, color, symbol, and line style, as well as the flags for contouring, breaklines, stringing, and annotation.

When data is imported and reduced in MAGNET Office, the code defined for each survey point is compared with the entries in this library and the appropriate attributes are allocated to the points and lines created by the dataset.

The survey codes are located in a library available for use with all projects. The Survey Code Library displayed is found in the “SvyCodes.lcd” file in the library defined on the **Libraries** tab (**Settings** ► **Program Settings**). Ideally, these settings should be common to all users.

When setting up the Survey Code Library, you should have already created symbols and line styles to allocate to the survey codes, as well as a decision on the colors that are applied for the various codes.

Survey codes are important for editing data within a database. Survey codes allow you to preassign point characteristics during the data

collection process that simplifies and speeds the processing required when passing from data collection to data presentation.

A survey or feature code is a group of characters assigned to observations taken by the surveyor in the field. It is used to label the point with a descriptive remark. The main reasons for using survey/feature codes are to:

- assign attributes to each point
- control automatic stringing of line features, such as fences
- assign attributes to each string line created
- create and assign layers in a project
- identify individual surveyed points
- control whether a point is used in the surface model
- define a line feature as a breakline

Survey codes and their various commands will save you time in data reduction and plan preparation.

The delimiters used for the command code and comments can be defined by the user in the **Code Settings** command in the **Library** menu. Table 16-1 describes command code delimiters.

Table 16-1. Command Code Delimiters

Default Delimiter	Delimiter Function
* (asterisk)	Asterisk used as the command code delimiter
/ (forward slash)	Forward slash used as the comments delimiter
. (decimal point)	Decimal point used as the offset delimiter

The Six Parts of the Code

A code consists of six parts — string number, contourable flag, survey code, command code, offset code, and comment — all joined together in a single string with special characters used as delimiters. The string can be letters, numbers, or alphanumeric characters, depending on the capabilities of different data recorders. MAGNET Office works with all types.

1. String Number The string number is used to identify points as belonging to the same line or string. String numbers are optional for alphanumeric data recorders. The string number consists of anywhere between zero and three digits and may be designated to the front (prefix) or the rear of the code (suffix). For example, the first and second string entered with two digits might appear as:

01CODE	CODE01	02CODE	CODE02
prefix	suffix	prefix	suffix

With a two-digit string number, the maximum number of strings for any one survey code is 99.

2. Contourable Flag The contourable flag determines if a point is used in creation of a DTM or surface model for producing contours or for computation. This setting is optional for alphanumeric data recorders, but essential for numeric data recorders. The contourable flag has three options:

- 0 (Zero) not contourable – the flag is off
- Contourable – the flag is on
- Use the survey code setting for the contourable flag

You can position the contour flag code as either a prefix or a suffix to the string number. The advantage of using alpha characters for codes is that the contour flag can be missed out if the setting matches that defined for the code in the Library.

3. Survey Code Some data recorders use only numeric characters, while others use a combination of letters and number as alphanumeric characters. MAGNET Office accepts any of these formats.

Survey codes are stored in the library files where they are assigned attributes and properties such as layer, color, and line style. Any incoming code matching the MAGNET Office code will automatically assign those attributes and properties.

4. Command Code Command codes are optional codes used to automatically close strings, create arcs, form rectangles, scale symbols, and perform back sight checks. They must be preceded by a normal survey code and separated from it by a delimiter character.

Such command codes as **AS** for Arc Start, **AE** for Arc End, **C** for close string, **R** for rectangle, or **CHK** to denote a check measurement may be assigned within the **Code Settings** command in the **Library** menu.

Arc Start and Arc End

The nominated characters are used to signify the start and end of an arc. MAGNET Office allows for the computation of two and three point arcs, as well as compound arcs using these command codes. Other points can be recorded between the points that have these special command codes. Command codes can be combined with normal codes, but should appear after the code, and be separated from normal codes with a delimiter code.

In the case of two point arcs, the two coded points are treated as tangent points of the arc. The point immediately before the start of the arc and the point coded as the start of the arc, define the tangent bearing of the first straight. The point coded at the end of the arc and the next recorded point define the tangent bearing of the second straight.

In the case of a three-point arc, the start point coded ***AS**, the next point (survey code only) and the third point coded ***AE** create a three-point arc.

In the case where there are a number of points on the string between the start point of the arc and the end point of the arc, MAGNET Office creates a three-point arc between the start and end point, using the penultimate point on the arc as the third point.

In the case of compound arcs, the three-point method is used. You should record Start of Arc, second point, Start of Arc, second point, End of arc. The second Start of Arc point will be used as the common tangent point between the two arcs.

Close String

The nominated characters are used to close the string from the point recorded with **CODE*C** back to the first point on the string, such as: **03FCE*C**.

Rectangle

Three points of this feature are surveyed with the third point recorded with ***CODE*R***. The fourth point is created to form a four-sided polygon. The two previous points on the string and the third point for rectangles are used to intersect and create the final point. The new lines will be parallel to the existing lines, therefore, forming a parallelogram.

The new point created will be coded as ***CORNER*** and will have no level. When three points are used for the definition of the rectangle, the final shape is automatically squared off.

You can also use two points on a string and an offset with the second point recording this detail, such as: ***BLD*R3.5***. A rectangle is created between these two points and a parallel line at the offset specified.

Scale

The nominated character is used upon reduction to automatically set the *X* and *Y* scale for a point; therefore, if the code being used has a symbol set in the Survey Code list, its *X* and *Y* scale will be set accordingly, such as ***TR*S4.56***. This TR point has the appropriate symbol applied with an *X/Y* scale of 4.56. The scale measures the radius, not the diameter.

Check Shot

This option only works in the Neutral Editor. It will not work with the manual entry options or importing data from XYZ or PTS files.

The chosen code is used to define an observation that is nominated as being a check measurement to a point of known coordinates, such as a back sight check during a series of observations from a station.

ID=34...CO=STN*CHK

When this point reduces, it will check against the coordinates of point number 341 in the MAGNET Office database and provide you with a comparison that will be written in the Neutral File Reduction Report. If point 341 does not exist in the database, then you will be prompted to enter the coordinates for the point.

Contour

This option is useful when the contour digit is not being recorded alongside the string number. The chosen code is used after the delimiter character to turn off the contour flag for this point, such as ***0IFCE*Z***.

When the data is reduced, this point will have the Contourable attribute unselected. This is useful when the level of the point is not a reliable indication of a surface value.

Height

This option is useful when you do not want the recorded elevation to be annotated on a plan. The chosen code is used after the delimiter character to turn off the height annotation flag for this point, such as: ***0IFCE*Y***.

When the data is reduced, this point will have the Height Annotation attribute unselected, even if the survey code has this attribute selected as the default setting. This is useful when the level of the point is not a reliable indication of a surface value. It may be used with the previous code, such as: ***0IFCE*YZ***.

Offset String

This option is designed to create a parallel string to be generated from a single surveyed string. This enables you to survey one side of a string, such as a path, hedgerow or wall and have both sides of the path, hedgerow or wall coordinated.

5. Offset Code When it is not possible to observe directly to a point, an observation can be made to a remote position offset from the required point and a code entered to account for the offset. Offsets are measured from the observed point to the required point in any combination of three axes:

- Height (H) – positive up, negative down
- Tangential (T) – positive right, negative left
- Radial (R) – positive further away, negative closer

The delimiter in this case is the decimal point, which instructs MAGNET Office that what follows is an offset code. The delimiter only needs to be used once after which a combination of offsets can

be used. The axis of offset is defined by the single letter code — H, T, or R — and the direction of offset as either positive or negative. Enter only the minus sign (-) for negative distances, but DO NOT enter the plus (+) sign as this may be misread as a delimiter.

Code.T-0.5R0.3 – This places the required point 0.5 to the left of the observed point and 0.3 further away from the observed point, relative to the observer.

Code.H1.2T0.6 – This offsets the required point 1.2 m above and 0.6 to the right of the observed point (Figure 16-2).

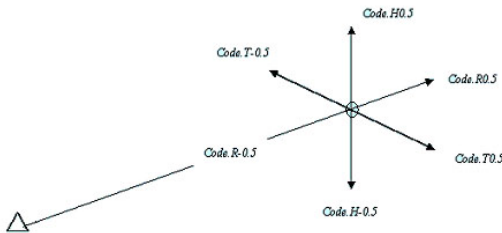


Figure 16-2. Offset Codes and Direction

6. Comments You can attach comments to a code for later reference, but comments are not used in any calculation or reduction. Comments provide additional information, such as details of a tree, species, trunk diameter, or height and spread of branches. They may also be used to apply a number to a feature, such as identified applied to manholes on different service lines.

Survey Codes

To use Survey Codes, do the following:

1. Click **Library** ▶ **Survey Codes**.

The *survey code library* dialog box appears (Figure 16-3 on page 16-9).

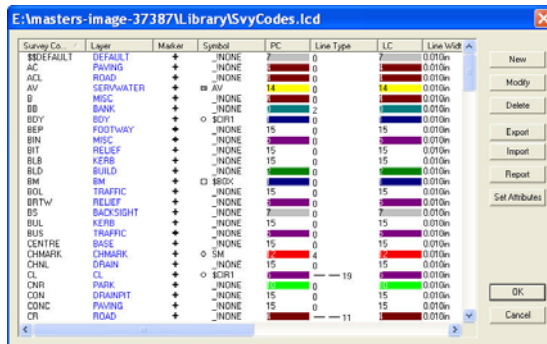


Figure 16-3. Survey Code Library Dialog Box

All of the survey codes in the library display with their properties and attributes in the columns on the left side of the dialog box, accessed via the horizontal scroll bar.

If a column header is clicked, the entities in the library will be sorted by this column.

The command buttons to change or create new survey codes are on the right side of the *survey code library* dialog box. Use the following to set parameters for survey codes.

- **New** – press to create a new survey code. The *Define New Survey Code* dialog box appears (Figure 16-4).

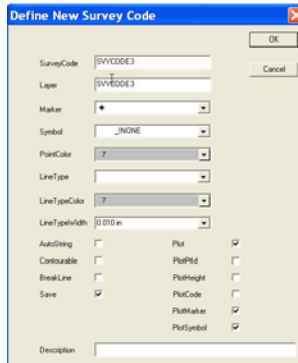


Figure 16-4. Create a New Survey Code



Select an existing code with similar settings to minimize the settings you need to change.

The following table (Table 16-2) describe the survey code field descriptions for the *Define New Survey Code* dialog box:

Table 16-2. Survey Code Field Descriptions

Field Name	Field Description
SurveyCode	Enter a name for the survey code.
Layer	Enter a layer name for the survey code.
Marker	Select a marker from the drop-down entry box.
Symbol	Select a symbol from the drop-down entry box.
PointColor	Select a point color from the drop-down entry box.
LineType	Change the line style for the lines created between points with this code.
LineTypeColor	Change the line style color for the lines created between points with this code.
Line Type Width	Change the width of the line style for the lines created between points with this code.
AutoString	<p>Check mark this box to automatically string points with this code, crating lines between the points. This is used for points on line features, such as fence lines, banks, and curbs.</p> <p>Leave is unselected for point features, such as ground levels, trees, poles, signs, and survey marks.</p>
Contourable	<p>Check mark this box to set points with this code as contourable. This is used for points on the surface of the survey, such as banks, ground levels, and road features.</p> <p>Click to clear the check box for points that are not on the surface, such as underground services, eaves of buildings, and culverts.</p>
Breakline	<p>Check mark this box to set lines strung from points with this code as breaklines. This is used for points on the surface of the survey, which are joined by lines that indicate a break in the surface grade. This can be applied to banks, curbs, drains, streams, and walls.</p>

Table 16-2. Survey Code Field Descriptions

Field Name	Field Description (Continued)
Save	Check mark this box to store the point in the database when the data is reduced.
Plot	Not used.
PlotPtId	Check mark this box to activate the Number Annotation for the point when it is created.
PlotHeight	Check mark this box to activate the Height Annotation for the point when it is created.
PlotCode	Check mark this box to activate the Code Annotation for the point when it is created.
PlotMarker	Not used.
PlotSymbol	Not used.
Description	Enter a suitable description for this survey code.
OK	Press to save the changes and return to previous screen.
Cancel	Press to quit and return to previous screen.

- **Modify** – select the code you want to change, then press **Modify**. The *Survey Code Properties* dialog box appears (Figure 16-5).

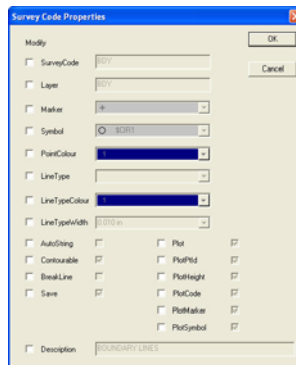


Figure 16-5. Modify a Survey Code

- To modify a code, check mark the box next to it and modify as needed.
- **Export** – press to export the survey code library to a TopSURV code file (*.xml) to use with TopSURV field software. In

addition, the library can also be exported to a GTS-700 code file (*.KDD), to use with the GTS-700.

- **Import** – press to import civilcad5 survey code files with the following extensions: *.BED and *.OUT.
- **Report** – press to create a report of all the survey codes and attributes, which you can either view on the screen and/or print or save to a report (*.rep) file.
- **Set Attributes** – To set the current attributes on the Property Toolbar, select one code and then press **Set Attributes**. The settings on the Property Toolbar are changed to use the attributes of the selected code. You can achieve the same results by selecting a feature code from the drop-down menu of codes on the right side of the toolbar.
- **OK** – press to save the survey code settings.
- **Cancel** – press to quit and return to the main screen.

Code Settings

Use the Code Settings command to define the various settings for controlling the stringing together of feature codes. It has other settings for command codes and delimiters used to extend the functions available when survey codes are used in the field. You should apply definitions to the codes on the **Survey Code Settings** dialog box to match the functionality available in the data collectors. Ideally, these settings should be common to all users of MAGNET Office.

To use the Code Settings command, do the following:

1. Click **Library ▶ Code Settings**.

The *Survey Code Settings* dialog box appears (Figure 16-6 on page 16-13).

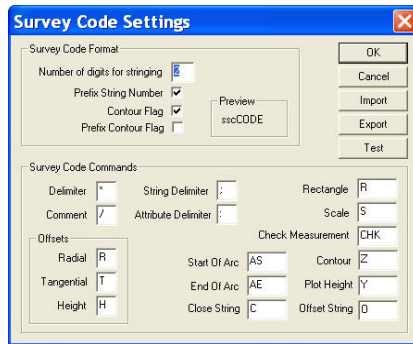


Figure 16-6. Survey Code Settings

2. Modify the properties as necessary, and then click **OK** to save your changes. For field descriptions, see Table 16-3.

Table 16-3. Survey Code Settings Field Descriptions

Field Name	Field Description
Survey Code Format	Sets the order in which the code, string numbers, and contour flags are entered in the instrument. The preview box displays the settings.
Number of digits for stringing	<p>Enter the number of digits used with the feature code for stringing. This is usually two, allowing 99 string numbers for any one feature code.</p> <p>The number of digits determine the number of different strings available using the same descriptive survey code. Using two digits for the string would provide as many as 99 different strings using the same survey code. A three-digit string provides 999 different strings using the same survey code.</p>
Prefix String Number	<p>Check mark this box if the string number prefixes the feature code. This is the usual setting - SScode such as: 03TB.</p> <p>Clear this check box if the string number is a suffix to the feature code - codeSS such as: TB03 or TB3.</p>

Table 16-3. Survey Code Settings Field Descriptions (Continued)

Field Name	Field Description
Contour Flag	<p>Check mark this box if the contour flag is entered with the string number. This is essential when numeric feature codes are used.</p> <p>This check box may be selected although the contour digit is not entered with the alpha feature codes. In this case, the contour flag is assumed to be set at 2= “read the contour flag from the library” for this feature code. Other contour flags used:</p> <p>0 = contour flag not set 1 = contour flag set</p> <p>Clear the check box if no contour flags are used. If no flag is used, then the default setting for the Survey Code is applied to the point.</p>
Prefix Contour Flag	<p>Check mark this box if the contour digit is a suffix to the string number. This is the usual setting. SSCcode such as: 031TB or 030TB.</p> <p>Clear this check box if the contour digit prefixes the string number CSScode such as 103TB or 003TB.</p>
Survey Code Commands	
Delimiter	<p>The default is * the asterisk. Another character should be chosen if the asterisk is not available on the data logger, such as + or #.</p> <p>The delimiter is used after the feature code to extend the survey code with further commands. The delimiter can indicate a second string and feature code to enable more than one string to be attached to a point, such as:</p> <p>02FL*04FL point at the intersection of two fence lines 02FL*06TB point on a fence line and a bank.</p>
Comment	<p>The default is the forward slash (/) . The space character may also be used.</p> <p>The comment character is used after the feature code to indicate that all characters that follow it are optional comments, such as Tree species, manhole number or a note - TR RIMU or TR/RIMU or SWMH 5A or SWMH/5A</p>
Start of Arc	<p>The default is AS, but may be modified as necessary. The character is used after the delimiter character to indicate the point is the start of an arc, such as: 03TK*AS</p>

Table 16-3. Survey Code Settings Field Descriptions (Continued)

Field Name	Field Description
End of Arc	<p>The default is AE, but may be modified as necessary. The character is used after the delimiter character to indicate the point is the end of an arc, such as: 03TK*AE</p> <p>When the data is reduced then a two- or three-point arc is created between the start and end points using another point on the string in between these points if available for the three-point arc.</p>
Close String	<p>The default is C but may be modified as necessary. The character is used after the delimiter character to indicate this point on the string in between these points if available for the three-point arc.</p>
Rectangle	<p>The default is R but may be modified as necessary. The character is used after the delimiter character on the third point of a rectangle. A new point is computed to form a rectangle with the other points, such as: 01BLD*R</p> <p>A new point will be computed at the fourth corner of the building and lines will join the first point on the string to the new point and from the labelled point to the new point. No level is computed for this new point.</p> <p>When three points are used, the rectangle is automatically squared off.</p> <p>You can also use two points and an offset such as: BLD*R3.5</p> <p>A rectangle is created between these two points and a parallel line at the specified offset.</p>
Scale	<p>The default is S but may be modified as necessary. The character is used after the delimiter character to set the symbol scale on the point. It is followed by a number representing the radial size of the point feature, such as: TRE*S3</p> <p>When the data is reduced, the point will have the symbol scale set to X scale = 3 and Y scale = 3.</p> <p>The symbol should usually be a scalable symbol.</p>

Table 16-3. Survey Code Settings Field Descriptions (Continued)

Field Name	Field Description
Check Measurement	<p>The default is CHK but may be modified as necessary. The character is used after the delimiter character to label a check shot to a back sight or other point with known coordinates, using the same point number already in the dataset.</p> <p>This usage only creates a check shot. It does not include the shot in the averaged calculations when more than one shot is made to one point, such as two shots on left and right face.</p>
Contour	<p>The default is Z but may be modified as necessary. The character is used after the delimiter character to turn off the contour flag for this point, such as: 01FCE*Z</p> <p>This is useful if the level shot in the field is not reliable and should not be used in a DTM.</p>
Plot Height	<p>The default is Y but may be modified as necessary. The character is used after the delimiter character to turn off the height annotation flag for this point, such as: 01FCE*Y. This can be used with the contour flag as 01FCE*YZ.</p>
Offset String	<p>These are used when it is not possible to observe the required point directly, but it is possible to offset the target by a known amount. Many instruments have these features on board, but these options can be used in the coding if the instrument does not compute offsets.</p>
Offsets	
Radial	<p>The default is R and is used after a decimal point, which directly follows the feature code with a positive or negative distance for the offset.</p> <p>For example: Code.R0.5 computes the point 0.5m further away from the observer than the target position and Code.R-1.00 computes the point 1m closer to the observer.</p>
Tangential	<p>The default is T and is used after a decimal point, which directly follows the feature code with a positive or negative distance for the offset.</p> <p>For example: Code.T0.5 computes the point 0.5m to the right of the observation at the target position and Code.T-1.00 computes the point 1m to the left of the observation.</p>

Table 16-3. Survey Code Settings Field Descriptions (Continued)

Field Name	Field Description
Height	<p>The default is H and is used after a decimal point which directly follows the feature code with a positive or negative distance for the offset.</p> <p>For example: Code.H0.3 computes the point 0.3m above the observation at the target position and Code.H-0.5 computes the point 0.5m below the observation.</p>

Create Custom Colors

Use the *Create Custom Colors* dialog box to define the 16 basic colors (0-15) and additional non-standard colors using the custom colors palette.

To create a custom color, do the following:

1. Click **Library ▶ Colors**.

The *Create Custom Colors* dialog box appears (Figure 16-7).

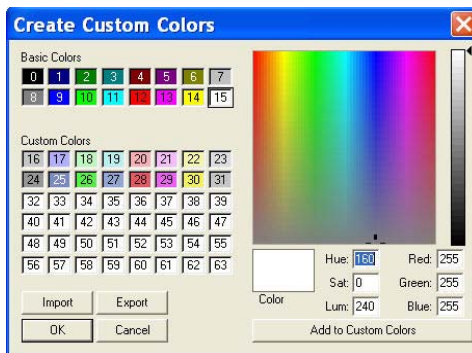


Figure 16-7. Create Custom Colors

2. Select a custom color number (24-63), and then a basic color (0-23).

The color options in this range display in the *Color* field on the right.

3. To customize a color, click in the shade bar or enter a numeric value in the field.

4. Press the **Add to Custom Colors** button to select and save a color to the selected number.
5. Press **Load from Lib** to create a color palette.
6. Press **Save from Lib** to save a color palette.
7. Press **OK** to save and recall a color palette.
8. Press **Cancel** to quit and return to the main screen.

Text Styles

Use the Text Styles command to define or modify text styles used in MAGNET Office. The text styles are located in a library available for use with all projects. The text styles displayed in the command are those found in the *Textstyle.lts* file in the library defined on the **Libraries** tab of the *Program Settings* dialog box (**Settings** ▶ **Program Settings**).

1. To open the Text Styles Library, click **Library** ▶ **Text Styles**.
The *text styles library* dialog box appears, listing all of the available text styles in the library (Figure 16-8).

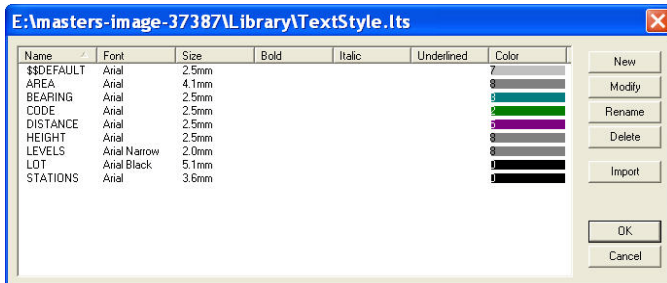


Figure 16-8. Text Styles Library

To define or modify text styles used in your project, do the following:

2. **New** – press to open a new text style. The *TEXT STYLE* dialog box appears. (Figure 16-9).



Figure 16-9. Add a Text Style

3. Enter a name for the text style, and select a **Font**. A preview of the font type appears in the *Preview* box below.
4. Select a font **Size** from the drop-down entry box.
5. Select a **Color** from the color palette drop-down entry box.
6. From the *Style* panel, select a style for the font type, either Bold, Italics, or Underline.
7. Click **OK** to add the new style to the library
8. Click **Cancel** to exit the *text styles library* dialog box and return to the main screen.



The style is added to the list of text styles. It is not added to the library of text styles.

Modify

To modify a text style, do the following:

1. Select a style that you want to modify, and click **Modify** on the *text styles library* dialog box.

The **TEXT STYLE** dialog box appears with the current style settings (Figure 16-10).



Figure 16-10. Text Style Window

2. Modify the properties as necessary, and then click **OK** to accept the new settings.

Rename

To rename a text style, do the following:

1. Select the text style from the Text Styles Library, and click **Rename**. The name becomes highlighted.
2. Type a new name, and then click **OK** to accept the change.

Delete

To delete a text style, select it from the Text Styles Library and click **Delete**. The style is deleted from the Text Styles Library but not removed from the project if the style has already been used in the project.

Copy From Library

To import a text style, press the **Copy from Lib** button. You can copy or import a defined text style in the program library into the current project.

Justification Styles

The MAGNET Office library contains user-defined number justification styles. When numbers are entered as text, annotation, or plotted in a project or drawing, the format of the number and its decimal precision is defined by a justification style. The style is embedded in the project, so the style is maintained, regardless of the library settings, when the project is opened at a later date.

This option lists the justification styles used in the project and those embedded from the default annotation settings when the project is created.

Use the Justification Styles command to define or modify a justification style used in MAGNET Office. Justification styles control the presentation and precision of numerical values when displayed as text in such items as height annotation, grid values, chainage, and levels in Roads drawings.

Justification styles are found in a library available for use with all projects. The justification styles displayed in this command are those settings located in the *Textstyle.lts* file in the library defined on the **Libraries** tab of the **Program Settings** dialog box.

To open the Justification Styles Library, click **Library ▶ Justification Styles**.

The *library textstyles* dialog box appears, listing all of the justification styles in the library (Figure 16-11).

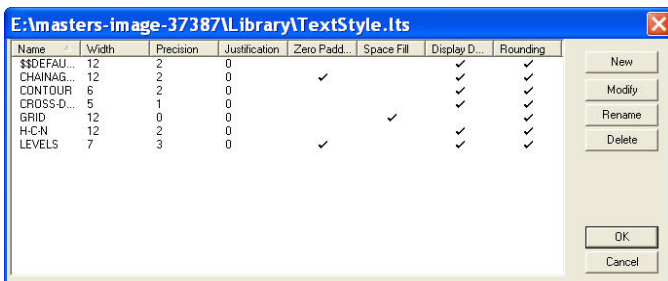


Figure 16-11. List of Justification Styles Library

Changes made to the justification styles in this list only apply to the library for use in new projects. The text styles already used in existing

projects are not changed. Text styles used in an existing project can be updated by clicking **Settings ▶ Justification Styles**.

Each text style is defined with a width, precision, justification position, or other option.

New

To create a new justification style for the library, do the following:

1. Select a current style that has similar settings to the new style, and click **New** on the *library textstyles* dialog box (Figure 16-11).

The *Number Style* dialog box appears (Figure 16-12). The fields in this dialog box are the same as the columns on the *library textstyles* dialog box in Figure 16-11 and as described in Table 16-4 on page 16-22.

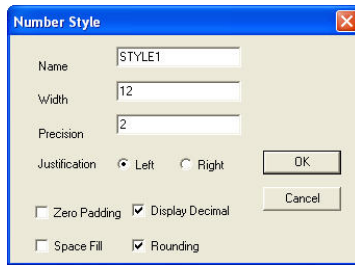


Figure 16-12. Number Style Window

2. Modify the properties as necessary to make a new style, and then click **OK**. The style is added to the library.

Table 16-4. Number Style Field Descriptions

Field Name	Field Description
Name	Enter a name for the new justification style.
Width	Enter the number of digits used in the style. This number includes the characters used for the decimal point and decimal figures.
Precision	Enter the number of decimal places in this style.
Justification	Select to justify the numerical text to the Left or Right .

Table 16-4. Number Style Field Descriptions

Field Name	Field Description
Zero Padding	When the justification is set to the Right , the displayed number is prefixed with zeroes to fill the character spaces in the width. For example, if the width is 8, the precision is 2, and the height is 12.00m, then Zero Padding will display as 00012.00.
Space Fill	When the justification is set to the Right , the displayed number is prefixed with spaces to make up the number of character in the width. For example, if the width is 8, the precision is 2, and the height is 12.00m, then the display is 12.00 where there is a space filling the character spaces before the number. This option is ideal for positioning chainage and level values in profiles.
Display Decimal	Indicates that the decimal point is displayed. If the precision is set to 0, then the decimal point will be omitted if this check box is not selected.
Rounding	Indicates the number is rounded to the precision setting.

Modify

To modify a justification style, do the following:

1. Highlight a style that you want to modify, and click **Modify** on the *Justification Styles Library* dialog box.

The *Number Style* dialog box appears with the current style settings (Figure 16-13).

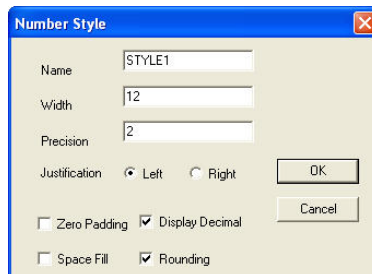


Figure 16-13. Number Style Window

2. Modify the properties as necessary, and then click **OK** to accept the new settings.

Rename

To rename a justification style, do the following:

- Select the style from the Justification Styles Library, and click **Rename**. The name becomes highlighted.
- Type a new name, and then click **OK** to accept the change.

Delete

To delete a justification style, highlight it from the Justification Styles Library and click **Delete**. The style is deleted from the Justification Styles Library but not removed from the project if the style has already been used in the project.

Copy From Lib

To import a justification style, highlight a justification style and press **Copy from Lib**. See “Text Styles” on page 16-18 for more information.

Annotation Styles

The **Annotation Styles** command in the **Library** menu is very similar to the **Annotation Styles** command in the **Settings** menu. For information about the *Annotation Styles* dialog box, see “Define Annotation Styles” on page 8-35 in Chapter 8.

Legal Description Templates

Use the Legal Description Templates command to create a legal description of a land parcel. The legal description templates in the library allows you to customize the legal description components. As

a result, the templates contain user-defined text and existing legal text to produce a customized legal description as shown in Figure 16-14.

John Doe Property

Jane Doe of the County of Fayette, State of Kentucky have Granted, Sold and Conveyed and by these presents do Grant, Sell and Convey unto the said John Doe of the County of Fayette, State of Kentucky all that certain piece or parcel of land situated in Harlan County, Kentucky, and more particularly described as follows: A tract of land containing 1.27 acres more or less, being part of the 5 acre tract conveyed to Mary James from Jesse Smith dated April 14th, 1989 and recorded in deed book 445, page 23 of the Deed Records of Harlan County Kentucky.

Commencing at a brass monument set in concrete, stamped NE corner Hickory Star Golf course, thence on a bearing of S51°20'25"E and distance of 101.491 ft to a Iron Pin (Set);
 Thence on a bearing of N88°45'17"E and distance of 104.184 ft to a nail;
 Thence on a bearing of S55°23'20"E and distance of 97.668 ft to a Iron Pin (found) PLS #3242 (KY), being the true point of beginning.
 Thence on a bearing of N73°41'54"E and distance of 250.072 ft to a Iron Pin (Found);
 Thence on a bearing of S27°56'43"E and distance of 210.183 ft to a Iron Pin (Set);
 from whence a USGS Mon. "Grey" located 1/2 mile west of the intersection of KY533 and KY38. bears S0°29'07"E a distance of 83.523 ft;
 Thence on a bearing of S74°57'21"W and distance of 296.603 ft to a brass monument stamped Hickory Star Golf Course Property Corner;
 Thence on a bearing of N15°08'28"W and distance of 199.389 ft to the point of beginning,
 said tract contains 55,309.326 Sq.Ft. or 1.27 Acres by survey made by Wise Man.

This conveyance is made subject to any and all easements of record in the office of the County Clerk of Harlan County, Kentucky effecting the lands herein conveyed.

Figure 16-14. Example of a Legal Description Generated from a Template

To define the template used to prepare a legal description, do the following:

1. Click **Library** ► **Legal Description Templates**.

The *Legal Description Template Library* dialog box appears (Figure 16-15).

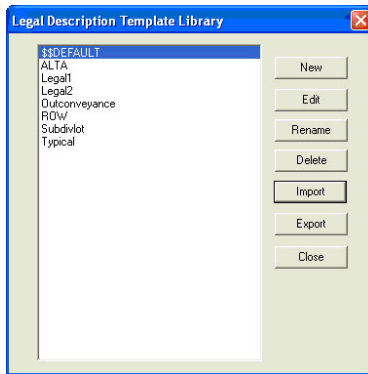


Figure 16-15. Legal Description Template Library

2. Highlight a template, and click **Edit**.

The component content displays on the *Legal Description Template Design* dialog box (Figure 16-16 on page 16-26).

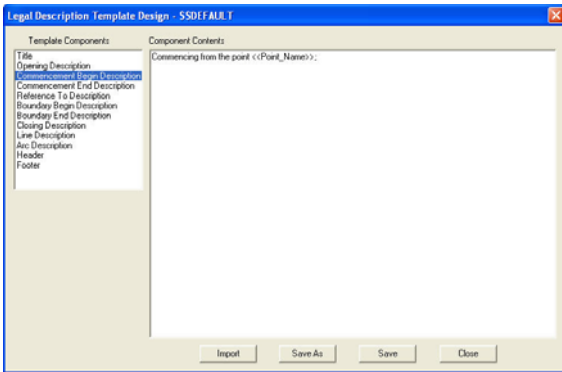


Figure 16-16. Component Content

3. Highlight the first component from the **Template Components** list to customize.

A description appears in the *Component Contents* text field. Each component has specific elements that can be defined. These elements are called keywords.

4. Enter the required information for the keywords. The keywords are identified by the double arrows << >>. In Figure 16-16, for example, enter the point name between the double arrows.
5. If necessary, add keywords to a component by right-clicking and selecting an option from the **Keywords** submenu. Your selection appears in the component contents. See “Keywords” below for more information.

Continue customizing keywords for each component in the **Template Components** list.

6. When you have finished, click **Save** to modify the template or click **Save As** to save the components as a new template that you name.
7. If you saved the components as a new template, the template name appears in the template library list.

Legal Description Template Library

For a description of the command on the **Legal Description Template Library**, see Table 16-5 below.

Table 16-5. Template Library Button Descriptions

Button	Description
New	Creates a new template.
Edit	Edits an existing template; see “Keywords” below for a description of the pop-up menu on the Legal Description Template Library screen.
Rename	Rename an existing template.
Delete	Deletes an existing template.
Import	Imports a template in .xml format.
Export	Exports an existing template in .xml format.
Close	Closes the dialog box and ends the command.

Keywords Keywords represent the element that will be inserted into the template at the selected insertion point. Each template component contains specific keywords that are available for insertion into the description. For example, Opening Description and Line Description contain different options as shown in Figure 16-17.

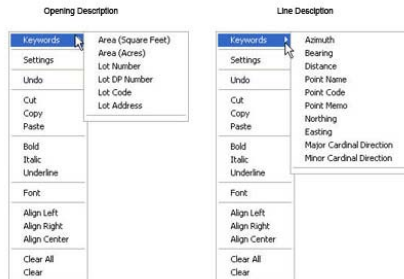


Figure 16-17. Keyword Submenus

To insert a keyword, do the following:

1. Place the pointer at the insertion point on the **Legal Description Template Library** dialog box.
2. Right-click and highlight **Keywords** to access the submenu.

3. Select an option from the **Keywords** submenu.
 The selected keyword will be inserted in strategic locations within the components.
 For a description of the complete list of right-click shortcut submenu options, see Table 16-6 below.

Table 16-6. Legal Description Template Library Menu Options

Shortcut Menu Options	Menu Option Descriptions
Keywords	Inserts the selected keyword at the insertion point. This keyword will be inserted at strategic locations within each component.
Settings	Opens the Legal Description Settings dialog box.
Cut Copy Paste	Standard Microsoft Windows editing functions.
Bold Italic Underline	Text options
Font	Provides font style, size, color, and other font options.
Align Left Align Right Align Center	Controls the text alignment position.
Clear All Clear	Clears all template components. Clears the active component only.

Define Template Components

Not all components need to be defined. Only define the components that are needed for the legal description. For template descriptions, see Table 16-7 below.

Table 16-7. Template Components Descriptions

Template Name	Description
Title	Describes the template.
Opening Description	Defines the opening description.
Commencement Begin Description	Defines the beginning point of a commencement path.

Table 16-7. Template Components Descriptions (Continued)

Template Name	Description
Commencement End Description	Defines the ending point of the commencement path.
Reference to Description	Defines the reference point description.
Boundary Begin Description	Defines the beginning point description.
Boundary End Description	Defines the description ending terminology.
Closing Description	Defines additional terminology required for closing statement description.
Line Description	Defines descriptive text for line bearings.
Arc Description	Defines descriptive text for curve bearings.
Header	The Header and Footer components are viewed in a word document (.rtf). The date and project name may be required. To add a keyword to the Header or Footer, select the Header or Footer component, right-click, and select the desired keyword.
Footer	

Symbols

Symbols are the defined shapes attached to points or survey marks that represent particular point features on the printed plan. Symbols may be scalable to represent the true size of a feature on a plan (tree, manhole, power box, etc.) or nonscalable to plot at a specified millimeter size on the plan (old iron tube, fire hydrant, etc.)

The Symbols commands allow you to compile or modify symbols stored in the library and used in MAGNET Office. The library symbols are available for use with all projects. The symbols displayed in this command are located in the *Symbol.lsy* file in the library defined on the **Libraries** tab of the program settings (**Settings ▶ Program Settings**).

To use the Symbols command, click **Library ▶ Symbols**.

The *Symbols Library* dialog box appears (Figure 16-18 on page 16-30).



Figure 16-18. Symbols Library

All of the symbols in the library display. Changes made to the symbols in this list only apply to the library for use in new projects. The symbols already used in existing projects are not altered.

Add Symbols to a Library

To create a new symbol for the library, do the following:

1. Click **Library** ► **Symbols**.
2. On the *Symbols Library* dialog box, click **New** (Figure 16-19).
The Symbols View displays, and a default name (*SymbolLib1*) is set for the symbol.

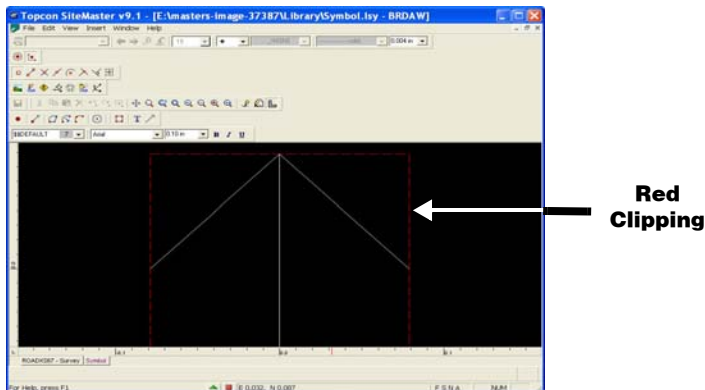




Figure 16-19. The Symbols View

3. To design a symbol, do the following:
 - Start with a point 0.0 as the center of the symbol or the position at which the symbol will attach to the point. Click **Insert ▶ Point** or click the **Point**  icon on the toolbar.
 - Create new points, lines, or arcs using the various options to represent the shape and size of the symbol. Make sure the line width is suitable for the symbol size.
 - Click **Insert ▶ Text** to add text.
 - Click the **Hot Button**  at the bottom of the dialog box. The *Definition* dialog box appears (Figure 16-20).

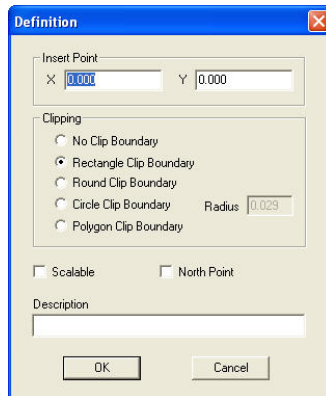


Figure 16-20. Defining Parameters for a New Symbol

- Click **OK** to complete the definition of the symbol.

For field descriptions, see Table 16-8. The clipping boundary is set with a red dotted line (Figure 16-19 on page 16-30).

Table 16-8. Definition Field Descriptions

Field Name	Field Description
Insert Point	Enter the X/Y values of the insertion point. The Insertion Point is usually X=0.0 and Y=0.0, but modify these values if you want to use another position.

Table 16-8. Definition Field Descriptions (Continued)

Field Name	Field Description
Clipping	<p>Select one of the options below that defines where the line features are clipped at the symbol when the plan is plotted:</p> <ul style="list-style-type: none"> • No Clip Boundary – Select this option when there is no boundary set for clipping. This is usually set for symbols for point features. • Rectangle Clip Boundary – Select this option to set the clipping at the four edges of a rectangular symbol. • Round Clip Boundary – Select this option to set the clipping edges of a circular symbol. • Circle Clip Boundary – Select this option to set the clipping at the circle circumference defined by the value in the Radius field. • Polygon Clip Boundary – Select this option to set the clipping at the extent of a polygon shaped symbol.
Radius	Enter a radius for the insertion point.
Scalable	Check mark this box if the symbol is a scalable symbol that represents a feature to scale on the plan, regardless of the plan scale used. If this check box is not selected, then the symbol will plot at the size created for all plan scales.
North Point	Check mark this box if the symbol will be used as a north point on the drawing. Most symbols have this check box unselected.
Description	Use this optional field to include a description of the newly created symbol.
OK	Saves the point information and returns to the main screen.
Cancel	Cancels the point information and returns to the main screen.

4. To save the new symbol, click **File ▶ Save** on the main screen.
5. To close the Symbols View, click **File ▶ Close** on the main screen.

Edit a Symbol


To edit an existing symbol, do the following:

1. Select the symbol from the Symbol Library (**Library ▶ Symbols**).

The *Symbol Library* dialog box appears.

2. Select a symbol, and then click **Open**.

The *Symbols View* screen appears, displaying the symbol design set within a red dotted box (Figure 16-21).

3. Modify the symbol using the Symbols View toolbars.
4. Complete the symbol definition by clicking the Hot Button  at the bottom of the dialog box.

The *Definition* dialog box appears (see Figure 16-20 on page 16-31).

5. Make the appropriate selections, and then click **OK**. For field descriptions, see Table 16-8.
6. To save the edits, click **File ▶ Save**.

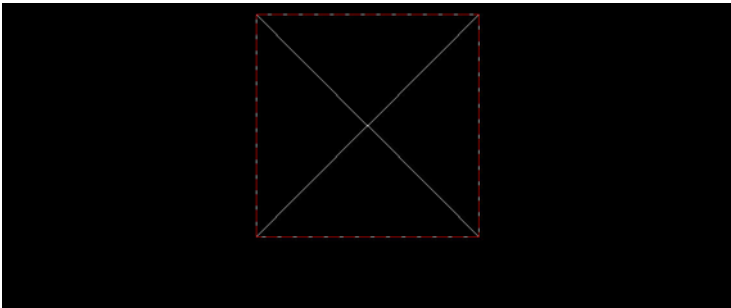


Figure 16-21. Symbol Design Displays Within the Red Dotted Box

Rename a Symbol

To rename an existing symbol or a new symbol with a default name:

1. Highlight the symbol from the list, and then click **Rename**.
2. Type the desired name, and then press **Enter** or click **OK** to accept the new name.

Delete a Symbol

To delete a symbol from the library, highlight the symbol from the list, and then press **Delete**. The symbol is removed from the Symbols Library.

Import a Symbol

To import a symbol to the Symbols library, press **Import**. The Open dialog box appears. You can import Civilcad5 symbol files (*.BCD) and/or AutoCAD drawingfiles (*.dwg) into the symbol library.

Line Styles

Line styles are defined styles used to represent particular line features on the printed plan. These line features may be standard solid, dashed, or dotted features or user-defined formats to represent features, such as fence lines, drainage lines, or services. You can apply line styles to lines, arcs, circles, polygons, and strings in a project to represent the type of feature on the finished plan.

The Line Styles command allows you to compile or modify line styles stored in the library and used in MAGNET Office. The line styles are located in a library available for use with all projects. The line styles displayed in this command are located in the *LineStyle.lls* file in the library defined on the **Libraries** tab of the program settings (**Settings ▶ Program Settings**).

To use the Line Styles command, click **Library ▶ Line Styles**.

The *Line Styles Library* dialog box appears (Figure 16-22 on page 16-35).

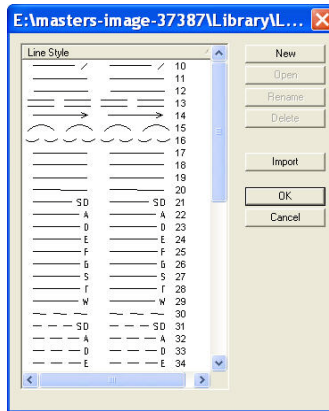


Figure 16-22. Line Styles Library

All of the line styles in the library display. Changes made to the line styles in this list only apply to the library for use in new projects. The line styles already used in existing projects are not altered.

Create a Line Style

To create a new line style, do the following:

1. Click **Library** ► **Line Styles**.
2. On the *Line Styles Library* dialog box, click **New**. The *Line Styles* dialog box appears (Figure 16-23).

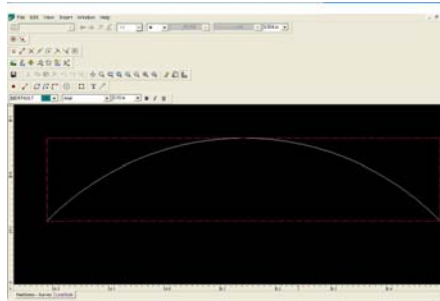




Figure 16-23. Line Styles Screen

3. To design a line style, do the following:
 - Start with a point 0.0 as the left end of the line style or the position at which the style will represent the line. Click **Insert ▶ Point** on the menu bar or click the **Point**  icon.
 - Create new points, lines, or arcs using the various options to represent the shape and size of one repeat of the line style, using spaces where they may be required. Design the line style left to right horizontally across the view.
 - Click **Insert ▶ Text** to add text if needed.
 - Click the **Hot Button**  at the bottom of the dialog box. The **Definition** dialog box appears (Figure 16-24).

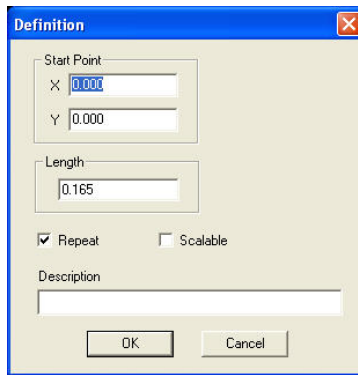


Figure 16-24. Defining Parameters for a New Line Style

4. Complete the line style definition; then click **OK**.
 For field descriptions, see Table 16-9 below.
 The clipping boundary is set with a red dotted square line.

Table 16-9. Line Style Field Descriptions

Field	Description
Start Point	Enter the X and Y values of the start point. The Start Point is usually X=0.0 and Y=0.0 but modify these values if you want to use another position.
Length	Set the length of one repeat of the line style. This may include a space before the next repeat of the line style begins.


Table 16-9. Line Style Field Descriptions (Continued)

Field	Description
Repeat	Select the check box to ensure the line style is repeated along the length of the line it is used for. If this check box is not selected, then only one repeat is used and the remainder of the line will be solid.
Scalable	Select this if the line style is scalable. If this check box is not selected, then the line style will plot at the size created for all line styles.
Description	Use this optional field to include a description of the newly created symbol.

5. To save the new line style, click **File ▶ Save** on the menu bar.
6. To close the Symbols View, click **File ▶ Close** on the menu bar.

Edit a Line Style

To edit an existing symbol, do the following:

1. Select the line style from the Line Style Library (**Library ▶ Line Style**).
The *Line Style Library* dialog box appears.
2. Select a style, and then click **Open**.
The *Symbols View* dialog box appears, displaying the line style design set within a red dotted box.
3. Modify the line style using the **Symbols View** toolbars.
4. Complete the line style definition by clicking the Hot Button  at the bottom of the window.
The *Definition* dialog box appears (Figure 16-25 on page 16-38).

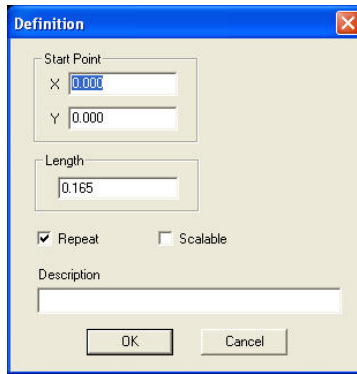


Figure 16-25. Defining Parameters for a New Symbol

5. Make the appropriate selections, and then click **OK**. For field descriptions, see Table 16-9.
6. To save the edits, click **File ▶ Save**.

Rename a Line Style

To rename an existing line style or a new line style with a default name, do the following:

1. Select the line style from the list, and then click **Rename**.
The name is highlighted.
2. Type the required name, and then press **Enter** or click **OK** to accept the new name.

Delete a Line Style

To delete a line style from the library, select the style from the list, and then click **Delete**. The line style is removed from the Symbols Library.

Import a Line Style

Press **Import** to import Civilcad5 (*.BLN) line styles.

Plot Settings

Click **Library ▶ Plot Settings**. The *Library Plot Settings* dialog box appears (Figure 16-26).

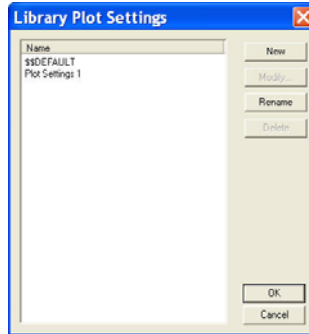


Figure 16-26. Library Plot Settings Dialog Box

Do the following to create, modify, rename, or delete a plot setting.

- **New** – press to create a new plot setting for the library. The *Library – Plot Settings* dialog box appears (Figure 16-28 on page 16-41). On the **General** tab set the following parameters:
 - Name: enter a name for the plot style.
 - Entities: check mark the entities you want to plot when using the current style.
 - Clipping: when toggled to **ON**, the entities outside of the border will be clipped to prevent them from being plotted.
 - Display World Coordinate: check mark this box to display real world coordinates of the points. If unchecked (toggled to **OFF**), you cannot identify point coordinates.



The coordinates can only be displayed and cannot be edited in any way within the plot window.

- Point Mark Plot Size (inch): enter a size to plot the point mark.

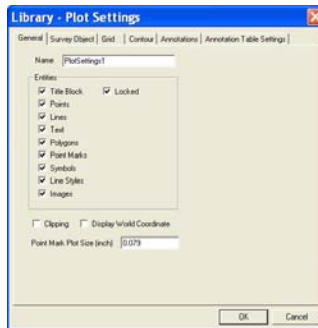


Figure 16-27. Library Plot Settings – General Parameters

On the *Survey Object* tab, select either *Grids*, *Contours*, *Triangles*, *Alignments*, or *North Point* for display on the screen.

On the *Grid* tab (Figure 16-28 on page 16-41), set the following parameters:

- Grid Marker – select one of the following from the *Grid Marker* panel:
 - Full Grid: select for grid to cover the entire page.
 - Grid Ticks at Border: select for grid ticks to appear only around the border.
 - Grid Ticks and Crosses: select for grid ticks and grid crosses to appear around the border and within the drawing.
- Spacing – this is in 1 to 1 units. If you want lines every 1/2 inch, set the spacing to 0.5. This is not determined by the plotting scale.
- Origin – enter East and North coordinates for the lower-left corner of the grid.
- Size – enter a value for the desired size for grid ticks and grid crosses.
- Text – select the following from the *Text* panel.
 - Left North: toggles ON and OFF to control coordinate labels.
 - Left East: toggles ON and OFF to control coordinate labels along the bottom of the border.

- Right North: toggles ON and OFF to control coordinate labels along the right side of the border.
- Right East: toggles ON and OFF to control coordinate labels along the top of the border.
- Above Ticks: controls the placement of the coordinate labels, either above or below the tick mark. When toggled to ON, the label is above the tick mark. When toggled to OFF, the label is below the tick mark.
- Justification Table – select a justification table from the drop-down entry box for the coordinate label.
- Text Style Table – select a text style table from the drop-down entry box for the coordinate label.
- Before/After North – enter an optional text label for coordinate values either before or after the north coordinate label.
- Before/After East – enter an optional text label for coordinate values either before or after the east coordinate label.



Figure 16-28. Grid Tab Options

On the *Contour* tab (Figure 16-29 on page 16-43), set the following parameters:

- Smoothing – check mark to create smooth contour lines in the drawing.
- Limit Smoothing – check mark to place a limit on the overlapping contours.

The result always depends upon the data in the DTM model. If unchecked, you may end up with curved contours that can overlap, especially in steep terrains and when contours are close together.

- Label – select the following label parameters:
 - Plot Labels: check mark to plot labels on major contours.
The result depends upon the other items selected.
 - Remove Overplotted Labels: check mark to ensure that contour labels are not overplotted by contour lines or other labels.
If check marked, there may be fewer contour labels that display in the drawing. Uncheck this box to display more labels. You may get better results by changing the label distance and reducing the text size.
 - Plot Labels Along Contour: check mark to fit the label into the shape of the contour wherever the label is positioned.
This results in each character of the label being individually plotted. The text in an unlinked drawing or an exported *.dwg/.dwf* file will be individual characters, instead of a text string, which makes it difficult to edit. Uncheck this box to ensure that the contour label is written as one text string.
 - Plot End Labels: check mark to write labels at the end of the contours wherever possible.
Labels will be omitted if the end of the contour deviates to the extent that there is no room for the label.
 - Plot Uphill Labels: check mark to display the contour labels so that they are read from the major contour uphill of the label. Uncheck this box to read the contour labels from the bottom or lower sides of the drawing area.
- Text Style – select a text style for the label(s).
- Justification Style – select a justification style for the display of the numbers in the contour label, which will include the length of the text string and the number of decimal places used.

- Label Distance – enter a distance in millimeters between repeats of major contour labels. Modifying this distance can give better results.



Figure 16-29. Library Plot Settings – Contour Parameters

On the *Annotations* tab (Figure 16-30 on page 16-44), set the following parameters:

- Point – check mark to display Name, Code, and Height parameters in the drawing.
- Line – check mark the *Bearing/Distance* parameter boxes to display on the table instead of adjacent to the line.
- Arc – check mark the *Radius, Chord, Length, Angle, or Chord Brg* parameter boxes to display on the table instead of adjacent to the line.
- Lot Annotation – check mark this box to have the lot annotation plotted in the table instead of adjacent to the line.
- Arc Annotation Table – check mark this box to control the position and format of the arc annotation table.
- Align Point Annotation with Sheet – check mark this box to modify the X/Y coordinate values of the Insertion Point.



Displaying parameters in a table is often advisable if a number of the arc elements are annotated, as there is rarely room to include all of these elements adjacent to the arc on the drawing.

- **OK** – press to save the settings and to close the Library - Plot Settings dialog box (Figure 16-30).

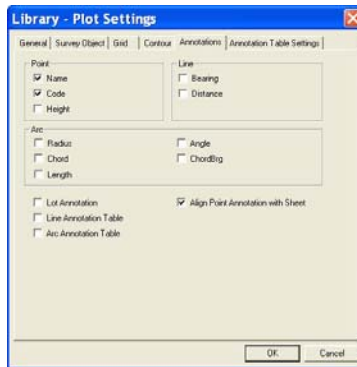


Figure 16-30. Library Plot Settings – Annotation Parameters

On the *Annotation Table Settings* tab (Figure 16-31 on page 16-45), set the following parameters for the annotation table:

- **Line/Arc Table Parameters** –
 - **Insertion Point X/Y:** Where the line/arc table will be placed by default. Doesn't really apply any more as it is now user defined on the screen.
 - **Columns:** enter the number of columns for the line/arc table. Typically 1 is sufficient. The line number, Bearing and Distance represents one column.
 - **ID Text Style:** Text style to be used for the line/arc table.
 - **Sorted By:** Select what line/arc table data will be used to sort the table.
 For example a line table can be sorted by either the line number, bearing or distance.

- Ascending: When checked, data will be sorted by the lowest value to the highest. If not check marked, the sort will be in descending order.
 - Display: Controls the visibility of the table. Toggle ON to display, toggle OFF to not display.
 - Reset: This option resets the table to the defined values specified in the **library > plot > annotation table settings**.
- Reset – press to set default values.
 - Local Number – this option sets each table to a sequential number method, beginning at Line or Arc number 1. This results in both the line and arc tables having local sequential numbers.
 - **Refresh** – will refresh existing line/arc tables with new settings if defined.

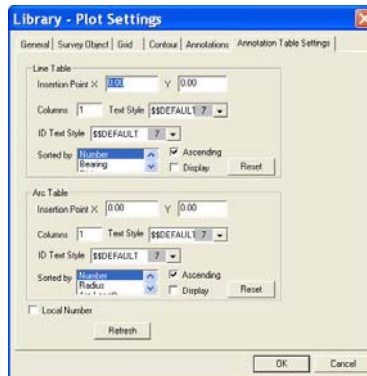


Figure 16-31. Library Plot Settings – Annotation Table Settings Parameters

- **Modify** – highlight a name on the *Library Plot Settings* dialog box (Figure 16-26 on page 16-39), then press **Modify** to edit a plot setting. The *Library – Plot Settings* dialog box appears (Figure 16-27 on page 16-40). Select parameters from the General tab menu options.
- **Rename** – highlight a name, then press **Rename** to change the name of a plot setting (Figure 16-26 on page 16-39).
- **Delete** – highlight a name, then press **Delete** to remove the name of a plot setting (Figure 16-26 on page 16-39).

- **OK** – press to save a library plot setting and return to the main screen (Figure 16-26 on page 16-39).
- **Cancel** – press to exit without saving changes.

Contour Settings

Click **Library ▶ Contour Settings**. The *Library Contour Settings* dialog box appears (Figure 16-32).

Do the following to create, modify, rename, delete, or copy a contour setting.

- **New** – press to create a plot setting for the library. The *Contour Settings* dialog box appears (Figure 16-32).
- **Modify** – highlight a name, then press **Modify** to edit a plot setting. The *Contour Settings* dialog box appears (Figure 16-27 on page 16-40).
- **Rename** – highlight a name, then press **Rename** to change the name of a plot setting.
- **Copy** – highlight a name, then press **Copy** to copy the settings of an existing style to make a new one.



If an existing style has most of the settings you want, use the Copy command to create a new style, then edit only the items you want to change. Select **NO** to the first prompt.

- **OK** – press to save a library plot setting and return to the main screen.

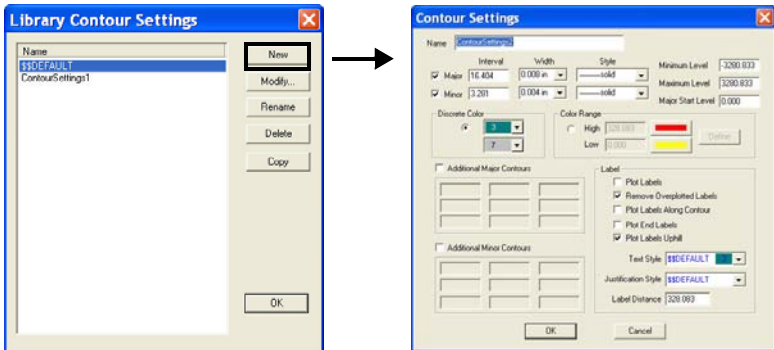


Figure 16-32. Library Contour Settings

Contour Settings Parameters

Figure 16-32 and Table 16-10 below describes contour settings parameters.

Table 16-10. Contour Settings Parameters

Contour Field	Contour Description
Name	By default, the first name for the contour setting is set as ContourSettings1. Additional contour settings will be named, ContourSettings2, ContourSettings3, etc. You can rename if you want to something else.
Major	see Discrete Color contour field.
Minor	see Discrete Color contour field.
Interval	Select an interval for the contour from the drop-down list.
Width	Select a width for the contour from the drop-down list.
Style	Select a style for the contour from the drop-down list.
Minimum / Maximum Level	Enter the range of levels in which the contours are created. The range should be between the lowest and the highest point of elevation in the DTM.
Major Start Level	Enter the level where the first major contour is created.

Table 16-10. Contour Settings Parameters (Continued)

Contour Field	Contour Description
Discrete Color	Select this option to open the color palette and set a Major and Minor line color.
Color Range	
	<ul style="list-style-type: none"> • High/Low: Press Define (the <i>Color Range Settings</i> dialog box appears) to select the highest/lowest color range or accept the value for the highest/lowest contourable point in the project. • Press the Color button to choose a color from the Color palette screen. <p>The heights/lowest DTM elevation displays with the color you selected.</p>
Additional Major/Minor Contours	In addition to the levels calculated by MAGNET Office, you can select either or both check boxes and specify the levels where the extra contours are created.
Label	Check mark the label type you prefer: either Plot Labels, Remove Overplotted Labels, Plot Labels Along Contour, Plot End Labels, Plot Labels Uphill.
Text Style	Select the text style you want for the contour.
Justification Style	Select the justification style you want for the contour.
Label Distance	Enter a parameter for distance.
OK	Press to save parameter settings and return to Library Contour Settings dialog box.
Cancel	Press to quit and return to the <i>Library Contour Settings</i> dialog box.

Genio Translation Settings

Genio files can basically have cross-section data, horizontal alignment data, and vertical alignment data. It is a generalized input/output format used to import/export various design file formats.

Click **Library** ► **Genio Translation Settings**. The *Genio Translation Tables* dialog box appears (Figure 16-33).

Do the following to create, modify, rename, save, or delete a Genio Translation Table (Figure 16-33).

- **New** – creates a new translation table. The *Moss Export LookUp Table* dialog box appears (Figure 16-34).
- **Modify** – highlight an entity, then press **Modify** to edit a translation table. The *Moss Export LookUp Table* dialog box appears (Figure 16-34 on page 16-50).
- **Rename** – renames a highlighted entity name from the table.
- **Delete** – removes a highlighted entity name from the table.
- **OK** – save a Genio Translation setting.
- **Cancel** – quits and exits without saving changes.

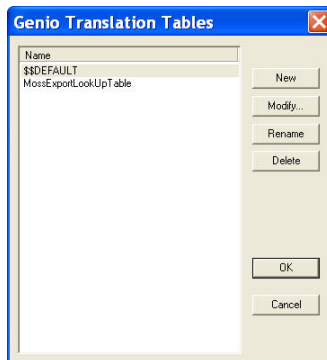


Figure 16-33. Genio Translation Tables

Figure 16-33 on page 16-49 and Table 16-11 describes the Genio Translation fields.

Table 16-11. Genio Translation Requirements

Field Names	Field Description
Table Name	By default, the first Genio Translation table name in a project is set as is MossExportLookUpTable1. Additional table names will be named, MossExportLookUpTable2, MossExportLookUpTable3, etc.
Entity	Enter an entity name or the table.

Table 16-11. Genio Translation Requirements

Field Names	Field Description
Code/Name	Mainly used if you have already created MAGNET Office strings or alignments and named them. These MAGNET Office strings will be converted to a specified Moss string.
Layer	Allows you to select a layer from within the MAGNET Office project to be converted into the specified Moss string name.
Moss String	Specifies the Moss String names required for MAGNET Office strings, alignments, or layers.
Insert	Press to insert an additional row before the currently highlighted row in the table.
Delete	Press to delete the currently highlighted row from the table.
Restore	Press to restore the table to the settings before items were edited.

When data entry is complete, press **OK** to save the new table parameters or press **Cancel** to quit.

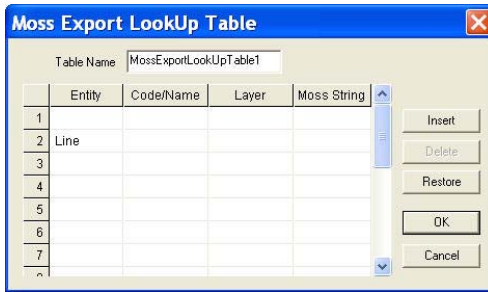


Figure 16-34. The Moss Export LookUp Table Parameters

Autocad Layer Definition Tables

Use the Autocad Layer Definition Tables command to set up a table of layer names and layer settings (color, line style, and thickness) to use in the Autocad Layer Translation table for *.dxf* and *.dwg* file exportation.

You can create new or edit existing definition tables in the library.

To use the Autocad Layer Definition Tables command, do the following:

1. Click **Library ▶ Autocad Layer Definition Tables**.

The *Layer Definition Table* dialog box appears (Figure 16-35).

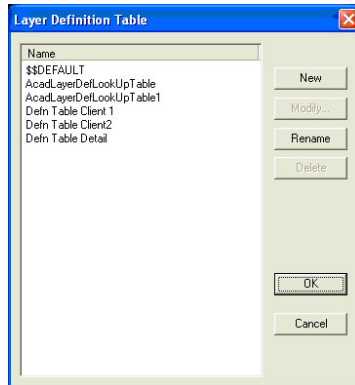


Figure 16-35. Layer Definition Table

2. To delete a table, highlight it, then click **Delete**.
3. To rename a table, highlight it, then click **Rename**. Type a new name and press **Enter**.
4. To edit an existing table, highlight it; then click **Modify**. On the second **Layer Definition Table** (Figure 16-36), make the necessary modifications; then click **OK** to save your changes. For a description of the *Layer Definition Table* fields, see Table 16-12 on page 16-52.

	Autocad Layer	Colour	Line Style	Thickness
1	0	15	solid	0.10 mm
2	BANK	3	solid	0.25 mm
3	BUILD	2	solid	0.25 mm
4	CL	5	19	0.25 mm
5	EB	20	solid	0.60 mm
6	FENCE	3	10	0.25 mm
7	NS	1	solid	1.00 mm
8	POWER	4	12	0.25 mm
9	TRACK	2	solid	0.25 mm
10	TREE	10	16	0.25 mm

Figure 16-36. Layer Translation Table

5. To create a new table, click **New**. On the *Layer Definition Table* dialog box, define the necessary settings, and then click **OK** to create the table.
6. When you have finished, click **OK** on the **Layer Definition Table**

Table 16-12. Layer Definition Table Field Description

Field Name	Field Description
Table Name	Displays the table name.
Insert	Creates a new row above a selected row. To select a row, click on the row number.
Delete	Deletes all selected rows from the table. To select a row, click on the row number.
Restore	Restores the original settings. No changes are saved.
Get Job Layers	Retrieves all layers and the layer settings from the current project.
OK	Accepts and saves all new changes to the table.
Cancel	Exits the dialog box without saving changes.

Autocad Layer Translation Tables

Use the Autocad Layer Translation Tables command to specify which Autocad layer to translate in place of the MAGNET Office layer used in the project.

To use the Autocad Layer Translation Tables command, do the following:



Make sure the **Use Layer Translation Table** box is check marked on the *Exporting Autocad Options* dialog box before using this command.

The **Layer Translation Table** (Figure 16-37 on page 16-53) contains two columns. The *Layer* column lists the layers in the MAGNET Office project, the *Acad Layer* (Autocad) column lists the corresponding layer the MAGNET Office layer will be translated

into. You can also select which autocad layers to translate using the *Acad Layer Table* drop-down menu, which lists the layers defined in the Layer Definition Table.

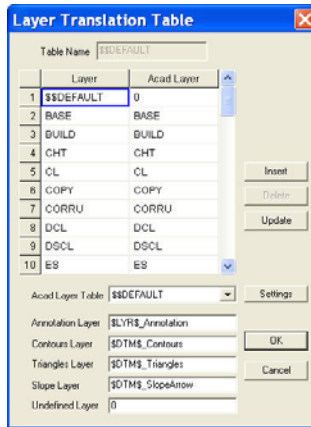


Figure 16-37. Layer Translation Table

1. Click **File** ► **Export**. Select a file, and then click **Save** to open this dialog box.
2. Click **Library** ► **Autocad Layer Translation Tables**.
The second *Layer Translation Table* appears (Figure 16-38 on page 16-54).
3. To delete a layer, highlight the row number; then click **Delete**.
4. To rename a layer, highlight it; then click **Rename**.
5. Type a new name; then press **Enter**.
6. To edit an existing layer, highlight the row number; then click **Modify**.
7. On the **Layer Translation Table**, make the necessary modifications; then click **OK** to save your changes. For a description of the **Layer Translation Table** fields, see Table 16-13 on page 16-54.

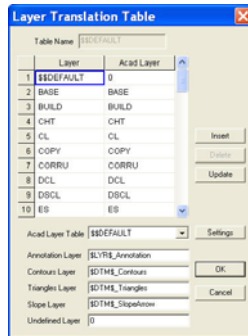


Figure 16-38. Layer Translation Table

8. To create a new layer, click **New**.
9. On the *Layer Translation Table* dialog box, define the necessary settings, and then click **OK** to create the layer. For a description of the **Layer Translation Table** fields and buttons, see Table 16-13.
10. When you have finished, press **OK** on the **Layer Translation Table**.

Table 16-13. Layer Translation Field Description

Field Name	Field Description
Acad Layer Table	Lists layer tables from the library.
Annotation Layer	All annotations are placed into the nominated layer name. As a default, they are placed in \$LYR\$_Annotation, where \$LYR\$ is the layer name of the entity that is annotated. All short line/arc tables are placed into the default layer 0.
Contours Layer Triangles Layer Slope Layer	All contour lines, triangles, and slope arrows are placed into the nominated layer. As a default, they are placed in \$DTM\$_Contours, \$DTM\$_Triangles, and \$DTM\$_SlopeArrow layers respectively.
Undefined Layer	Defaults to layer 0. All entities not specified to acad layers in the Translation table are placed
Insert	Creates a new row above a selected row. To select a row, click on the row number.
Delete	Deletes all selected rows from the table. To select a row, click on the row number.

Table 16-13. Layer Translation Field Description (Continued)

Field Name	Field Description
Update	Updates the table to include all of the layers in the current project.
Settings	Opens the Layer Definition Table .
OK	Accepts and saves all new changes to the table.
Cancel	Quits without making changes.

Page Setup

The Page Setup command is unavailable unless the data in the Survey View is a 2D project saved from a drawing, using the **File ▶ Save** command in the Drawing View.

Use the Page Setup command to fine-tune the drawing margins to ensure the project will plot on the chosen printer/plotter or to modify the sheet size. The title block used for the drawing, now a project, is not changed with this command.

To use the Page Setup command, click **Library ▶ Page Setup**.

The *Page Setup* dialog box appears (Figure 16-39).

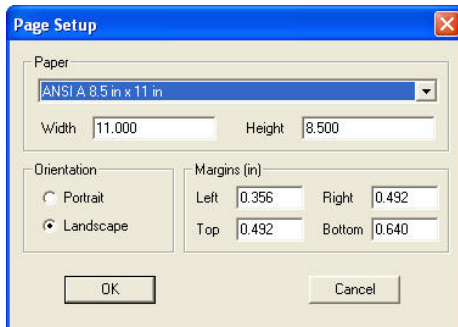


Figure 16-39. Page Setup Dialog Box

Only change these settings if the 2D project is not printing correctly on the printer/plotter specified in the **File ▶ Print Setup** command. Make sure the printer properties are correctly defined before making any changes in **Page Setup**.

For more information about the Page Setup command and dialog box, see “Properties Toolbar” on page 18-16.

Title Block

Use the Title Block command to save a designed plan form as a MAGNET Office title block. The title block defines a drawing, at a specified scale, position, and rotation using the **Library ▶ Title Block** command.

Design the title block in a project in the Survey View using the units as millimeters for all of the lines. Any text style can be used in the title block at the sizes currently defined in the Text Styles Library. Make sure the proposed scale set in the *Hot Button* dialog box is 1:1000.

The title block should include the border on the sheet and any title information, which is standard to all drawings. Points with symbols, lines with line types, arcs, circles, strings, and text may be used to build up the frame of the plan form and insert detail in the title areas of the plan form.

You can insert keywords with the correct text style and their position defined. These keywords display as *\$keyword* and are replaced by the appropriate text string set in the *Project Details* tab on the *Project Settings* dialog box (**Settings ▶ Project Settings**) when the project is created.

Make sure the extents of the plan form will print on the specified sheet size on the printer/plotter. Each printer/plotter model has its own minimum margins between the edge of the paper and the plotted extents of data. When the design is completed, save the project for future reference or editing of the title block.

To use the Title Block command, do the following:

1. Click **Library ▶ Title Block**.

The *Page Setup* dialog box appears (Figure 16-40).

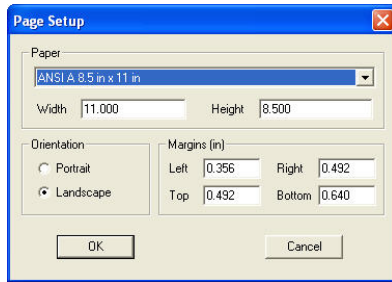


Figure 16-40. Page Setup Dialog Box

2. Set the paper size, orientation, and appropriate margins; then click **OK**.

A rectangular window (solid yellow line with inner dashed green lines) displays attached to the mouse in the upper left corner of the plot or drawing area of the plan form.

3. Click the origin point as stated on the prompt line at the lower-left side of the dialog box or enter the **East** and **North** coordinates in the input bar.

The yellow and green window are now fixed at this position.

4. Select the first point for defining the drawing area or enter the coordinates in the input bar.
5. Drag the upper-right corner of the pointer to the second point, creating a yellow rectangle, and click this position.


This defines the area in which the project data is plotted. It becomes the extents of the window when the plot window is defined using **Tools ▶ Create Plot Window**.

6. In the *Save As* dialog box, make sure the **Library** folder is opened, type a name for the title block, and then click **Save**. The title block is now saved for use in all projects.

Create Plot Window

To check the title block, do the following:

1. Click **Tools ▶ Create Plot Window** to use the title block to create a drawing around some data in a sample project.

2. In the Drawings View , click **File ▶ Print Preview** to preview the plot.
3. Make sure the title block appears as expected and fits correctly at 100 percent scale with the borders complete.
4. If it appears as expected, click **File ▶ Print** and draw a plan on the printer or plotter.

Setting Margins for the Title Block

If the title block is not plotting correctly with the defined margins, then make sure the correct printer is selected and the other settings, including the paper size, are suitable for the designed title block.

1. Select **Library ▶ Title Block**.
2. Complete the page setup, and set all of the margins to 0.00.
3. Click **OK** to display a message similar to the one in Figure 16-41.

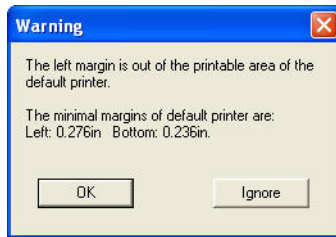


Figure 16-41. Warning Message

This message displays the minimum margins used by the printer or plotter at the lower left side of the paper.

4. Press **OK** to return to the *Page Setup* dialog box and make modifications.

Press **Ignore** to continue and set the margins to 0.00.

5. Save the title block and create a test drawing.

When the project is printed, the printer or plotter automatically sets the margins to the current minimum settings available within the printer.

Some plotters may have an option to set smaller margins than the default margins, however, this setting may be a hardware setting and not set in the printer driver.

MAGNET Office uses the printer or plotter set in **Print Setup** and the properties defined in its printer driver. Modify the properties of this driver to ensure correctly positioned plots.

Additional Information

- If you must modify the margins for every drawing, then you can update the margin settings in the title block file for all future drawings.
- Some plotters have rotation settings for maximum paper usage. This is especially obvious when plotting A2 title blocks rotated on roll feed paper. If the A2 title block is difficult to fit on the sheet, it may be possible to get better results if the paper size in the plotter driver is set to something slightly bigger than ISO A2. For example: try oversize A2 or Architecture C size.

Symbol

If you have drawn a symbol in the Survey View, then use the Define Symbol command to save it to the Symbol Library or to the current project.

To use the Define Symbol command, do the following:

1. Select the symbol using a selection window before saving the symbol.
2. Click **Library ▶ Define Symbol**.
3. Click on a point in the symbol data to which the symbol is attached.

The *Symbol Name* dialog box appears (Figure 16-42 on page 16-60).



Figure 16-42. Symbol Name Dialog Box

4. Enter a name for the symbol, and select one of the following:
 - Overwrite Job Symbol – overwrites an existing project symbol with a new definition.
 - Add to Symbol Library – adds the new symbol to the Symbol Library.
5. Click **OK**.

The *Definition* dialog box appears (Figure 16-43).

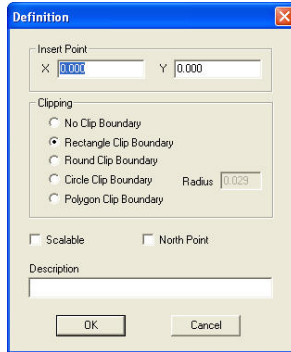


Figure 16-43. Definition Dialog Box

6. Define the clipping parameters and other options as required, and click **OK** to save the symbol.

Define Line Style

If you have drawn a line in the Survey View, then use the Define Line Style command to save it to the Symbol Library or to the current project.

To use the Define Line Style command, do the following:

1. Select the line before saving it.
2. Click **Library ▶ Define Line Style**.
3. Select an insertion point by clicking on a point at the left end of the line style.

The *Line Style Name* dialog box appears (Figure 16-44).

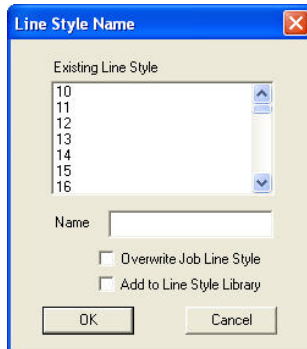


Figure 16-44. Line Style Name Dialog Box

4. Enter a name for the line style, and select one of the following:
 - Overwrite Job Line Style – overwrites an existing project line style with a new definition.
 - Add to Line Style Library – adds the new line to the Line Style Library.
5. Click **OK**.

The *Definition* dialog box appears (Figure 16-45 on page 16-62).

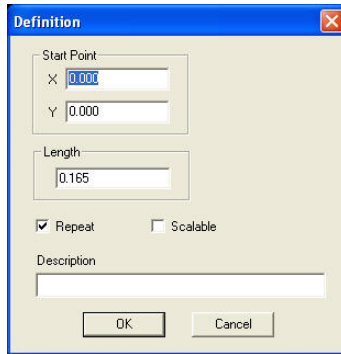


Figure 16-45. Definition Dialog Box

6. Define the length and other options as required; then click **OK** to save the line style.

Survey View: Window Menu/Help Menu

This section describes the different Survey View menu commands on the **Window** menu located on the main menu bar (Figure 17-1).

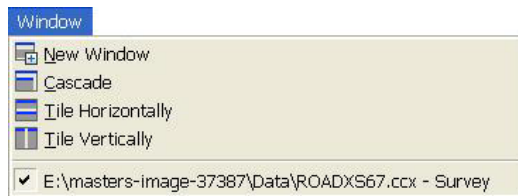


Figure 17-1. Window Menu Options

In This Section:

- “New Window” on page 17-1
- “Cascade” on page 17-2
- “Tile Horizontally” on page 17-3
- “Tile Vertically” on page 17-3
- “Help Menu” on page 17-4
- “Display Hints” on page 17-5
- “About Topcon MAGNET Office” on page 17-5

New Window

Use the New Window command to create a new window in the Survey View. Additional windows are useful if you want to work in one window that is mapped to one area of the project, as well as in another window mapped to a different area. Many routines in the

Survey View work across the windows, following the position and focus of the pointer.

To open a new window, click **Window ▶ New Window**.

A new window is created. The two windows are named **Survey:1** and **Survey:2**.

You can create additional windows and display them together using the “Cascade”, “Tile Horizontally” on page 17-3, and “Tile Vertically” on page 17-3 commands in the **Window** menu.

Cascade

Use the Cascade command to display all of the currently open windows as a series of one behind the other. The name of each dialog box is displayed on its title bar (Figure 17-2).

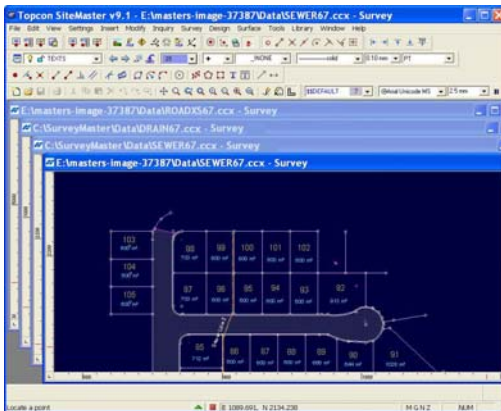


Figure 17-2. Cascading Windows

The active window is highlighted in blue on the title bar and will be the top window. You can move the windows to a new position by dragging them with the mouse.

If the cascaded windows are from different views within MAGNET Office, then the menu will change as a window becomes active.

Tile Horizontally

Use the Tile Horizontally to display one or more currently open windows one above the other to fill the working area of the screen (Figure 17-3).

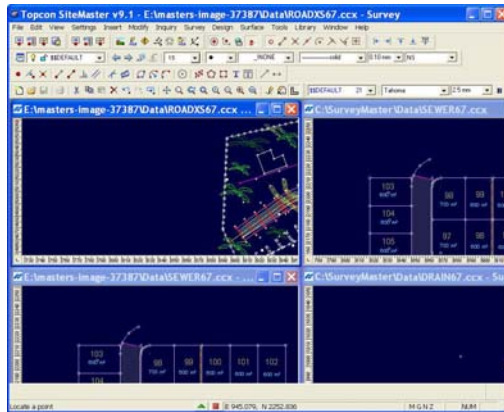


Figure 17-3. Tiled (Horizontally) Windows

You can move the windows to a new position by dragging them with the mouse. If the tiled windows are from different views within MAGNET Office, then the menu will change as a window becomes active.

Tile Vertically

Use the Tile Vertically command to display one or more currently open windows one beside the other to fill the working area of the screen (Figure 17-4 on page 17-4).

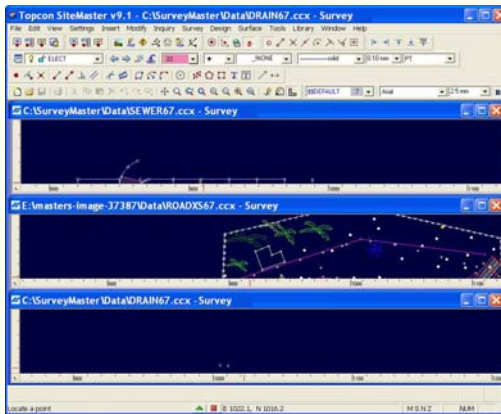


Figure 17-4. Tiled (Vertically) Windows

You can move the windows to a new position by dragging them with the mouse. If the tiled windows are from different views within MAGNET Office, then the menu will change as a window becomes active.

Help Menu

This section describes the different Survey View menu commands on the **Help** menu located on the main menu bar (Figure 17-5).



Figure 17-5. Help Menu Options

In This Section:

- “Help” on page 17-5
- “Display Hints” on page 17-5
- “About Topcon MAGNET Office” on page 17-5
- “Registration” on page 17-6

Help

Use the **Help ▶ Help** command to open the MAGNET Office Help files or press **F1**. These files are installed in the Help folder.

In the left pane, the Contents tab serves as a table of contents, listing all of the chapters in the Help system. These chapters are organized by “View,” which describes the MAGNET Office window you are currently looking at. These chapters are organized further into menu commands available in the View. To access these commands, click the + sign to the left of the folder. The + sign indicates there are additional folders and/or files in it. The - sign indicates there are no other folders or files to open.

When you find a topic you want in the left-pane table of contents, click it and the help contents is displayed in the right-pane window. Some topics may include links to other topics or instructions in the Help system that you can click on for additional information.

This Help system provides access to all instructional content available at the time this product is released.

Display Hints

Use the **Help ▶ Display Hints** command to view helpful tips that relate to the commands you are using.

The Display Hints window contains tips that apply to the task at hand. Just leave this window open and use it as a context-sensitive help reference.

About Topcon MAGNET Office

Use the **Help ▶ About Topcon MAGNET Office** command to display important information about the MAGNET Office software. You will need this information when making a service call or updating the software.

Registration

Use the **Help ▶ Registration** command to displays the *Program Registration* dialog box. If you have already registered your MAGNET Office program, your registration information displays in this dialog box (Figure 17-6).

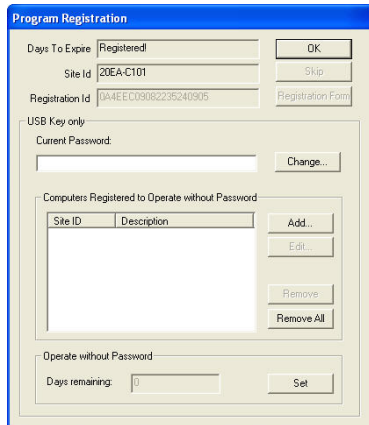


Figure 17-6. Program Registration Window

In this window, you can change your password, register computers to operate without a password, and registered your MAGNET Office program if you have not already done so.

Roads View Editor

The Roads View displays alignments, cross sections, and profiles in three separate panes.

You can move between the Survey View, the Drawing View, the Roads View, and the Raw Data Editor by clicking the appropriate icon on the **Modules** toolbar (Figure 18-1)

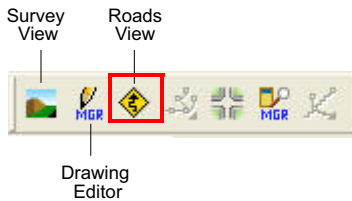


Figure 18-1. Cross Section Editor

Click on the *Road View* icon to Display the Roads View.

About the Road View Editor

When the Roads View Editor opens, three panes of data display on the main screen:

- Alignment Pane (left) – displays the alignment and design strings. The plan data from the Survey View may also be shown (Figure 18-2 on page 18-2).

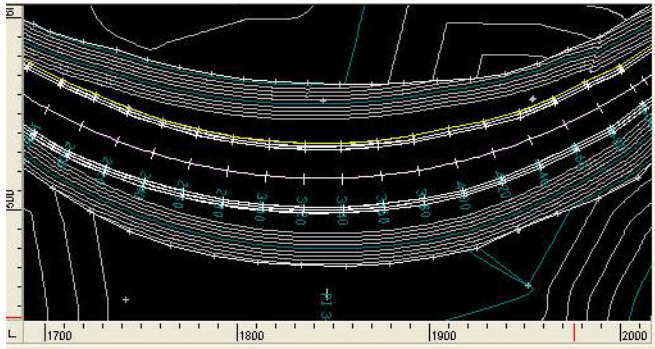


Figure 18-2. Alignment Pane

- Cross Section Pane (right) – displays the cross section for one chainage. Several surfaces may be shown (Figure 18-3).

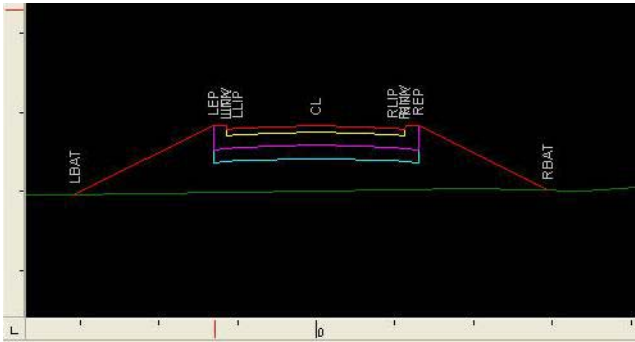


Figure 18-3. Cross Section Pane

- Profile Pane (bottom) – displays the profile for the natural surface and the design profile with vertical IP points for the reference alignment or any one selected string (Figure 18-4 on page 18-3).

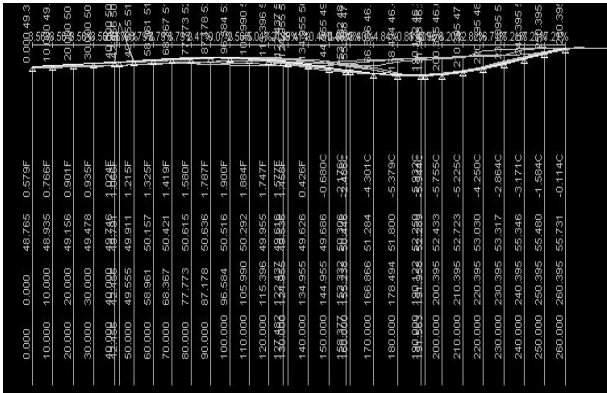



Figure 18-4. Profile Pane

The display in each window depends on the settings defined in the **Display Settings** (Hot Button ) dialog box located at the bottom of the window. See “Display Settings (Hot Button)” for more information.

Each pane has a ruler at the bottom. A short red line displays on these rulers to show the position of the pointer within the active pane.

- In the Alignment pane, X/Y coordinates display.
- In the Cross Section pane, offset and level parameters display.
- In the Profile pane, chainage and level parameters display.

In addition to the rulers, these pointer positions display on the Status Bar.

Like the Survey and Drawing Views, the Cross Section Editor has several toolbars and icons. To identify some of the Cross Section Editor features, see Figure 18-5 on page 18-4 and Table 18-1 on page 18-4.

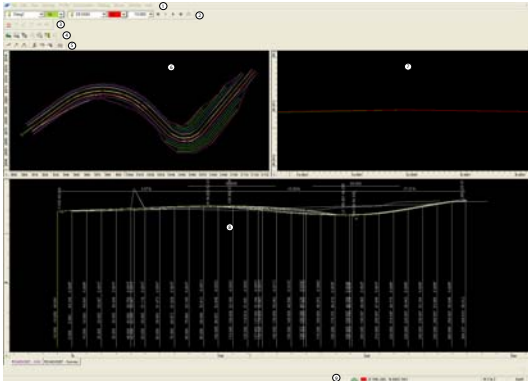


Figure 18-5. Cross Section Editor

Table 18-1. Window Description

Number	Description
1	Menu Bar
2	Road Property Toolbar
3	Design Toolbar
4	Modules Toolbar
5	Profile Toolbar
6	Alignment Pane
7	Cross Section Pane
8	Profile Pane
9	Status Bar

This chapter describes the different Cross Section Editor menu commands located on the main menu bar:

- “File Menu” on page 18-7
- “Edit Menu” on page 18-9
- “View Menu” on page 18-12
- “Settings Menu” on page 18-21
- “Profile Menu” on page 18-34
- “CrossSection Menu” on page 18-50

- “Plotting Menu” on page 18-107
- “Library Menu” on page 18-123
- “Window Menu” on page 18-135
- “Help Menu” on page 18-135


Creating Alignments

The alignment is the zero offset of the cross sectional data used for the road.

You can create a horizontal alignment from a line, a string, or an alignment frame with curve geometry applied at the horizontal intersection points along the alignment. Cross sectional data is extracted along the alignment from a natural or existing surface. This surface can be:

- A digital terrain model (DTM) created from surveyed or imported data, typically used in “green field” designs or subdivision development.
- Three-dimensional string data representing the changes in grade along individual feature lines, typically used when reconstructing or upgrading an existing road.
- Cross section point data surveyed at right angles to a pinned center line, usually at 10m or 20m spacings.
- Level book data entered into the **Survey ▶ Manual Data Entry ▶ Level Book** command in the Survey View.

Display Settings (Hot Button)

The *Display Settings* (Hot Button ) dialog box (Figure 18-6 on page 18-6) controls the screen display of certain parameters and is context sensitive. Context sensitive means that the page displayed when the Hot Button is opened depends on which pane is active. To make a pane active, click in that pane. To view the settings of a pane, select a pane from the **Settings** menu.

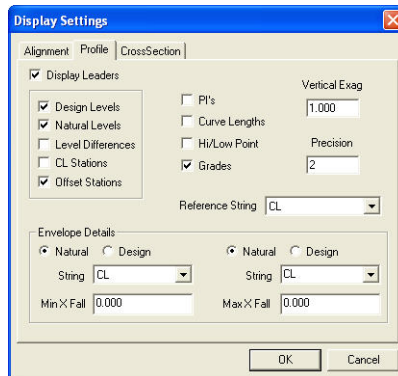


Figure 18-6. Display Settings (Hot Button)


Selecting a Current String

Left-click a stringline in the Alignment pane to make it the current string or select a string from the Properties toolbar. The current string name is displayed at the left end of the Properties toolbar. The current string is highlighted in yellow in the Alignment pane. Double-click on a string to display information or edit its properties.

Vertical Intersection Point

A Vertical Intersection Point (VIP) displays on the design profile in the Profile pane, with a triangular symbol representing the position of the VIP point. Its color is taken from the color applied to the current string on the Properties toolbar. Left-click a VIP symbol to select it or double-click a symbol to display the *Edit IP* dialog box for viewing or editing the parameters.

Displaying Leader Lines

There is a leader line at each chainage in the Profile pane. The relevant data displayed on the leader line is controlled by the settings in the *Display Settings* (Hot Button ) dialog box. See “Display Settings (Hot Button)” for more information. MAGNET Office hides the display of some leader lines if cross sections are very close

together and do not display well at the current mapped scale of the profile.

Double-click a leader line to display the details of the chainage, the design, the natural surface levels at this point, and the difference in levels. This is a useful tool when the displayed text on the leader line is not easy to read.

Viewing the Current Surface

The Properties toolbar includes a drop-down menu that sets one surface in the Cross Section pane as the current surface. This current surface has a mark at each offset point on the cross section. Double-click a mark to display details of the offset, level, code, and surface grades to either side of the mark. The code, level, or offset may also be displayed beside the mark.

File Menu

This section describes the different Cross Section Editor menu commands on the **File** menu located on the main menu bar (Figure 18-7).



Figure 18-7. Cross Section File Menu

In This Section:

- “Open” on page 18-8
- “Save” on page 18-8
- “Cross Section Editor” on page 18-9

Open

Use the Open command to open an alignment file and display it in a new window in the Cross Section Editor.

To open an Alignment file, do the following:

1. Click **File ▶ Open**.

The *Open* dialog box appears (Figure 18-8), listing the alignments in the current project.

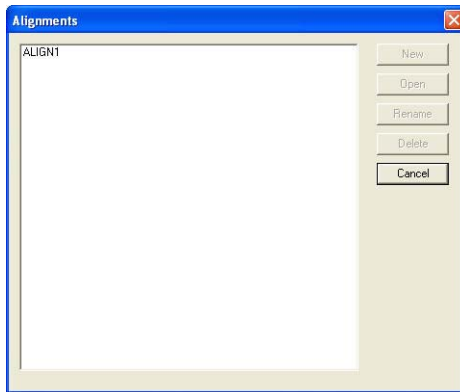


Figure 18-8. Alignment Dialog Box

2. Select the required alignment, and then click **Open**.

The alignment is opened in another dialog box of the Cross Section Editor, and this dialog box becomes the current view. The existing alignment remains open in its own dialog box.

Save

Use the Save command to save the current project and data in any currently open view of that project. When you use this command from within the Cross Section Editor, you are prompted to save any open datasets that have not already been allocated a name or have had the default name confirmed.

To save a drawing, do the following:

1. Click **File ▶ Save** or press **Ctrl+S**.

If an existing Road View has not been saved, then the **Save As** dialog box appears.

2. Select a folder in which to save the Road View, type a file name, and then click **Save**.

The edited data is saved. You are prompted to save other road views that have not yet been saved.

Cross Section Editor

To close the **Cross Section Editor** and return to the Survey View in the current project, click **File ▶ Close**. The Cross Section Editor closes and the Survey View opens, displaying the project in the current dialog box.



This command does not prompt you to save the road. The road is saved within the contents of the total project when the project is saved in the Survey View or **File ▶ Save** in the Cross Section Editor.

Edit Menu

This section describes the different Cross Section Editor menu commands on **Edit** Menu located on the main menu bar (Figure 18-9).

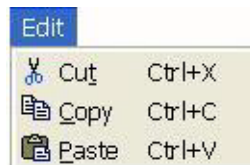


Figure 18-9. Cross Editor Edit Menu


In This Section:

- Figure on page 18-10
- “Copy” on page 18-10
- “Paste” on page 18-12

Cut

The Cut command removes the entities from the data set in a similar way to using the **Delete** command and places the cut data onto the clipboard.

To remove an entity from the data set, do the following:

1. Select the data or highlight it by dragging a window around it with the mouse, then do one of the following.
 - Click **Edit ▶ Cut**
 - Press **Ctrl+X**
 - Click the **Cut**  icon
2. To restore the cut data, click **Edit ▶ Undo**.
3. To paste the cut data from the clipboard to another position or into another drawing, click **Edit ▶ Paste**.
For more information, see “Paste” on page 18-12.


Copy


Use the Copy command to copy selected entities to the clipboard. The entities must be selected before this command becomes available. The selected data may be copied from one project and pasted into another at the same coordinates.

To copy a single entity to another position within the same MAGNET Office project, do the following:


1. Select the entity by clicking on it or dragging a window around it.
2. Press and hold the **Ctrl** key.
3. Click on the entity’s center (yellow) handle, and drag it to a new position.
4. Release the mouse to save the copied entity.

To copy a single entity to another project within MAGNET Office, do the following:

1. Select the entity by clicking on it or dragging a window around it.
2. Click **Edit ▶ Copy** or click the **Copy**  icon on the **Standard** toolbar.

3. In the other project in which you want to place the copied entity, click **Edit ▶ Paste** or click the **Paste**  icon on the **Standard** toolbar.
4. If you are pasting into another view, the *Duplicate Point Name Detected* dialog box appears. Check mark the **Renumber all points** check box (recommended) or choose to duplicate the point name, and click **OK**.
5. If you do not see the entity you pasted, zoom out by using the options on the right-click shortcut menu or the icons on the **Standard** toolbar.

To move an entity within the same project:

1. Select the entity by clicking on it or dragging a window around it.
2. Click **Modify ▶ Move**.
The entity becomes highlighted with a yellow box around it.
3. Click on the center point or handle.
The pointer changes to the **Move Entity**  pointer icon.
4. Drag the entity to a new position, or enter the new coordinates into the input bar and then press **Enter**.
The *Move Confirmation* dialog box appears (Figure 18-10).

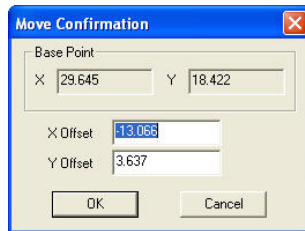


Figure 18-10. Move Confirmation Dialog Box

5. Click **OK** to place the entity into the new position.

You can copy text from one project and paste it into another project at the same coordinates. If the second project already contains data, the points will be renumbered to prevent overwriting existing points.

To copy and paste text from other compatible applications, do the following:

- Copy the text from the other application.

- Set the text style.
- Click **Insert ▶ Text ▶ Text**.
- Click **Edit ▶ Paste**.

If the text is pasted into MAGNET Office in one long text string (i.e. word wrap active in other applications) then double-click the inserted text and break the text string into multiple lines using the **Enter** key at suitable places in the text string.

Paste

Use the Paste command to paste data from the clipboard into the current project.

When the Paste command is used, entities are read from the clipboard and placed into the currently active window. You can drag the data to another coordinate position. The attributes and properties (coordinates, dimensions, color, line style, etc.) will remain the same.

The Paste command is only available if data was cut or copied to the clipboard. See also “Cut” on page 18-10 and “Copy” on page 18-10.

To paste data, either click **Edit ▶ Paste** or press **Ctrl+V**. The data on the clipboard is displayed in the dialog box.

Moving the pasted data: To move the pasted data:

1. Press and hold the **Shift** key.
2. Select the yellow handle on the lower left of the dialog box, and drag the handle and data to a new position.

View Menu

This section describes the different Cross Section Editor menu commands on the **View** menu located on the main menu bar (Figure 18-11 on page 18-13).



Figure 18-11. Cross Editor View Menu

In This Section:

- “Redraw”
- “Regen”
- “Zoom” on page 18-14
- “Pan” on page 18-14
- “Toolbar” on page 18-15
- “Status Bar” on page 18-20
- “Return to Survey View” on page 18-20

Redraw

Use the Redraw command to refresh the display on the screen.

To refresh the drawing display, click **View ▶ Redraw**.

Regen

Use the Regen command to regenerate the display of data on the screen by reading the project database.

To regenerate the data, click **View ▶ Regen**.

Zoom

Use the **Zoom** command to remap the data on the screen at a different scale.

To access the **Zoom** commands, click **View ▶ Zoom** or right-click on the mouse and choose an option from the shortcut menu (Figure 18-12).

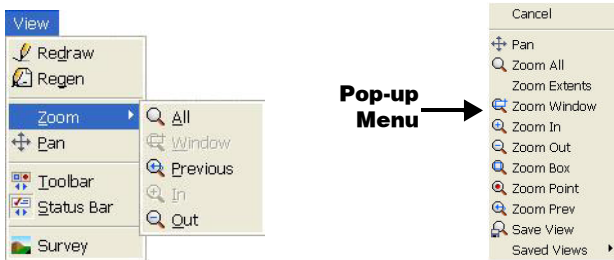


Figure 18-12. Zoom Commands

The **Zoom** commands in the Cross Section Editor are similar to the commands in the Survey View. For a description, see Table 7-1 on page 7-3 in the Survey View chapter.

Pan

Use the **Pan** command to move the data position in the display area without changing its mapped scale.

To Pan data, do the following:

1. Click **View ▶ Pan** or right-click and select **Pan** from the shortcut menu.
2. Click to define the first position.
3. Click to define the second position (the direction in which you want to move the view).

This moves the first point to the relative position of the second point.

4. Right-click, and select **Cancel** from the shortcut menu to finish the **Pan** command.



Press and hold the scroll wheel on the mouse at a chosen position and drag the pointer to a new position.

Toolbar

Use the **Toolbar** command to control the display of toolbars in the Drawing View. You can show or hide the toolbars, separate them, and reposition them anywhere in the dialog box vertically and horizontally by using the click and drag method.

To show or hide the toolbar, do the following:

1. Click **View ► Toolbar**. The *Configure Toolbars* dialog box appears (Figure 18-13 on page 18-16).
2. To hide a toolbar, click to clear the check mark next to it. The check mark indicates the toolbar is displayed on the screen.
3. To reset all toolbars to their default locations in every view, press **Restore Default**.
4. To arrange the toolbars inside or outside of the drawing area, click a toolbar (not an icon) and drag it to where you want to position it.
5. To save the position of the toolbars, press **Save Layout** on the **Configure Toolbars** page.
6. To restore the toolbars to their default positions, press **Restore Default**.

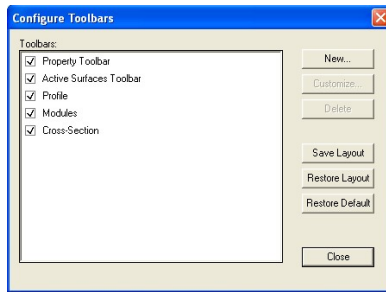


Figure 18-13. Configure Toolbars

Once you have moved a toolbar into the working area, you can also change the shape of the toolbar to fit more appropriately by placing the pointer over an edge of the toolbar until it changes to double arrows, and then clicking and dragging the toolbar edge.

The names of the individual tools on the toolbar are displayed when the pointer is moved over an icon.

For a description of the toolbars, see the following:

- “Properties Toolbar”
- “Active Surfaces Toolbar” on page 18-17
- “Profile Toolbar” on page 18-18
- “Modules Toolbar” on page 18-19
- “Cross-Section Toolbar” on page 18-19

Properties Toolbar This toolbar displays the current settings that are used for computing road design and displaying data in the three panes (Figure 18-14 on page 18-17). For a description of the toolbar, see Table 18-2 on page 18-17.

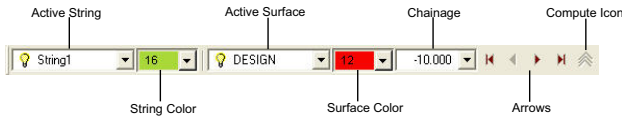


Figure 18-14. Properties Toolbar

Table 18-2. Property Toolbar Description

Toolbar Name	Toolbar Description
Active String	Selecting the correct active string is important before you enter data into the various data entry screens.
String Color	The color used for the active string in the alignment pane, for the design IPs, and for the profile line in the LongSection panel. This color also sets the color of the string when written to the Survey View.
Active Surface	The selected surface is the active surface in the Cross Section pane.
Surface Color	Each surface in the cross section can be allocated a different color.
Chainage	The chainage of the cross section that displays. Any chainage can be selected from the drop-down menu.
Arrows	The four arrow icons move forward or backward through the cross sections, or through the start or end chainage.
Compute Icon	Computes the design.

Active Surfaces Toolbar This toolbar changes “natural surfaces” (if multiple surfaces are contained in the project) and “design trial surfaces” to allow for review of trial designs (Figure 18-15).

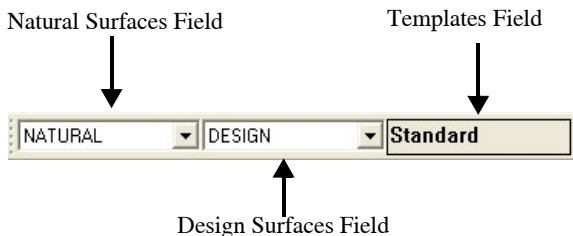
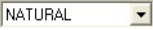




Figure 18-15. Active Surfaces Toolbar

For a detailed description of the active surfaces toolbar, see Table 18-3 on page 18-18.

Table 18-3 describes the toolbar fields.

Table 18-3. Active Surfaces Toolbar Description

Field Type	Field Description
	The Natural Surfaces field display the current natural surface (will only show one surface is only one natural surface was extracted) Otherwise, if multiple natural surfaces were extracted, you can select which natural surface to design to.
	The Design Surfaces field displays the current design surface that will display in Profile view. Changing to different design surface names in the pick list will update the Profile view display.
	The Template field (far right) relates to the template applied. If you have multiple template selections for a given road design, the “drop-down” chainage pick list will display the correct templates at any given chainage. If you use the horizontal left/right arrows to scroll through your design, the template display will be updated. If you snap a chainage leader line in Profile view, (displays the cross section at that snapped chainage) the template design will be updated.

Profile Toolbar Use these icons to design and edit the IPs in the profile design (Figure 18-16). For a description of the toolbar, see Table 18-4.



Figure 18-16. Profile Toolbar

Table 18-4. Profile Toolbar Icon Descriptions








Icon	Name	Description
	Add IP	Adds a vertical IP at a nominated chainage and level with an optional curve length.
	Add IP by Grade	Adds a vertical IP at a nominated chainage, at a level calculated by specifying a grade from another vertical IP.
	Add IP by Intersection	Adds a vertical IP at the chainage and level calculated by the intersection of grades from two existing vertical IPs.

Table 18-4. Profile Toolbar Icon Descriptions (Continued)

Icon	Name	Description
	Modify IP	Edits the parameters of an existing vertical IP, including specifying or calculating a suitable curve length.
	Move IP	Moves an existing vertical IP. The IP may be raised, lowered, or moved to another chainage. The profile line is displayed as the IP is moved.
	Delete IP	Deletes an IP.
	Snap to Chainage	Snaps the mouse to a leader line in the Profile pane.

Modules Toolbar You can move between the Survey View, Drawing View, Cross Section Editor, and Raw Data Editor by clicking the icons on the **Modules** toolbar (Figure 18-17).

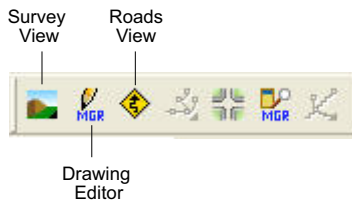


Figure 18-17. Modules Toolbar

Cross-Section Toolbar Press the Cross-Section toolbar  to update the design surface.

Status Bar

The status bar on the bottom of the dialog box provides information and several options. The Status Bar displays by default (Figure 18-18 on page 18-20). The check mark next to the **Status Bar** command in the **View** menu toggles on/off to show/hide the Status Bar at the bottom of the screen. To hide the Status Bar, click to clear the check mark in the menu. For a description of the Status Bar, see Table 18-5 below.




Figure 18-18. Status Bar

Table 18-5. Status Bar Description

Number	Description
1	This button only applies to the Survey View and Drawing View.
2	The red rectangle, also called the Hot Button, opens the Road Display Settings that control the display of data on the screen in three panes.
3	The rolling coordinates display the coordinates of the pointer position.
4	This displays the units that are set for the project.
5	This indicates that the Number Lock is activated.

Return to Survey View

To return to Survey View, click **View** ► **Survey** or the **Survey**  icon on the **Modules** toolbar in Drawing View.

The Survey View becomes the current dialog box. The windows for any Drawings open in the Drawings View will remain open.

Settings Menu

This section describes the different Cross Section Editor menu commands on the **Settings** menu located on the main menu bar (Figure 18-19).

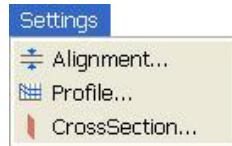


Figure 18-19. Cross Editor Settings Menu

In This Section:

- “Alignment Pane”
- “Profile Pane” on page 18-23
- “CrossSection Pane” on page 18-25

Alignment Pane

Use the Alignment command to control the display of data in the Alignment pane (Figure 18-20). These settings do not set the output format for plotting.

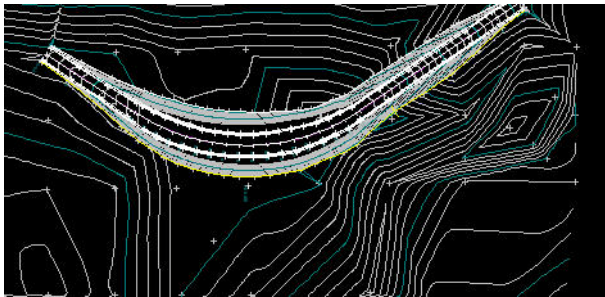



Figure 18-20. Display of Data in the Alignment Pane

To control the display of data in the Alignment pane, do the following:

1. Click **Settings** ▶ **Alignment** or click the Hot Button  at the bottom of the window.

The *Road Display Settings* dialog box appears (Figure 18-21).

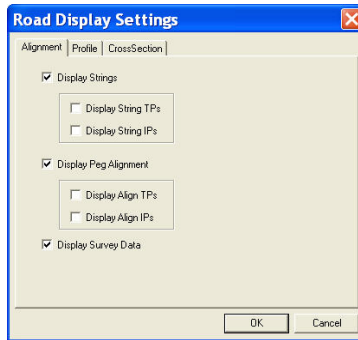


Figure 18-21. Display Settings – Alignment Tab

2. Make sure the *Alignment* tab is opened to modify the properties as shown in Table 18-6.
3. Click **OK** to close the dialog box and refresh the display on the screen.

Table 18-6. Alignment Field Descriptions

Field Name	Field Description
Display String	Displays the design strings after computation. You can assign a color to each string by making each string the current string and setting a color.
Display String TPs	Shows the tangent points on the design strings after computation. The tangent points display on the main design string.
Display String IPs	Shows the intersection points on every chainage of the design strings after computation.
Display Peg Alignment	Shows the alignment at the zero offset of the cross sections. The alignment uses the color set on the Properties toolbar for alignment.
Display Align TPs	Shows the tangent points on the alignment.
Display Align IPs	Shows the intersection points on the alignment.
Display Survey Data	Displays the data on active layers in the Survey View. If the triangular mesh and the contours are active in the Survey View, they will display in the Alignment pane.

Profile Pane

Use the Profile command to control the display of data in the Profile pane (Figure 18-22). These settings do not set the output format for plotting.

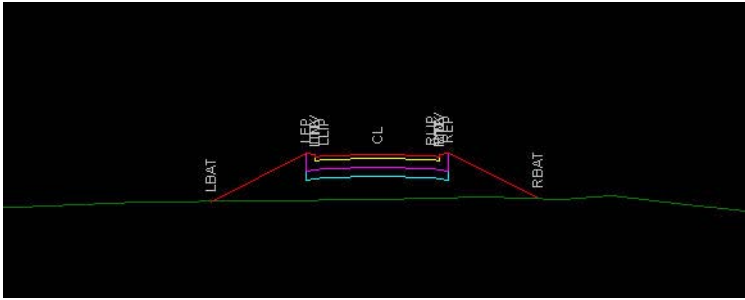



Figure 18-22. Display in the Profile Pane

To control the display of data in the LongSection pane, do the following:

1. Click **Settings** ► **Profile** or click the **Hot Button**  at the bottom of the dialog box.

The *Road Display Settings* dialog box appears (Figure 18-23).

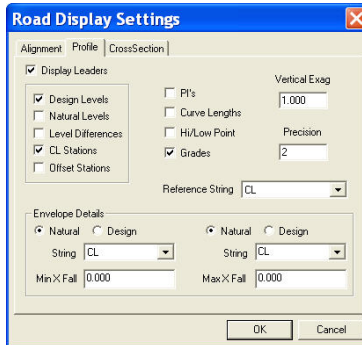


Figure 18-23. Display Settings – Profile Tab

2. Make sure the *Profile* tab is opened, and modify the properties as required in Table 18-7 on page 18-24.

- Click **OK** to close the dialog box and refresh the display on the screen.



The labelling of some leaders may be hidden if the mapping scale is set so that data overlaps.

Table 18-7. Profile Field Descriptions

Field Name	Field Description
Display Leaders	Displays the vertical leader lines on the screen.
Design Levels	Displays the design levels for the current string on each leader line.
Natural Levels	Displays the natural surface levels for the current string on each leader line.
Level Differences	Displays the difference between the levels on the two surfaces. Fill: Displays the distance the design surface is above the natural surface. Displayed as a positive value followed by F, such as 0.15F. Cut: Displays the distance the design surface is below the natural surface. Displayed as a negative value followed by C, such as -0.15C.
CL Stations	Displays the center line station
Offset Stations	Displays the station along the offset polyline or string.
PI's	Displays the levels of the intersection points (IP) for the current string. The chainage and level is displayed vertically above the triangular symbol used to define an IP position.
Curve Lengths	Displays the length of the vertical curves for each intersection point of the current string. The curve length is displayed at the top of the screen above a horizontal line that shows the position and length of the curve.
Hi/Low Point	Displays the high and low points on the design profile of the current string. The chainage and level is displayed vertically above the high or low point and marked HI for high point and LO for low point.

Table 18-7. Profile Field Descriptions (Continued)

Field Name	Field Description
Grades	Displays the grade of the designed road line between two adjacent intersection points (IP). The grade is displayed as a positive or negative percentage below the curve length data at the top of the screen and above a horizontal line that shows where the grades change at each IP point. The grade is measured from left to right, from the lower chainage or distance position to the higher chainage or distance position.
Vertical Exag	This value defaults to 1.000. Increase to change the vertical scale as a ratio to the horizontal scale in the on-screen display. A value of 2.000 sets the vertical scale at twice the horizontal scale.
Precision	This value defaults to 2 and sets the number of decimal places in the data displayed on the screen.
Reference String	Select the string used to define the chainages on the profile plot. This is not necessarily the current string.

CrossSection Pane

Use the CrossSection command to control the display of data in the CrossSection pane (Figure 18-24 and Figure 18-25). These settings do not set the output format for plotting.

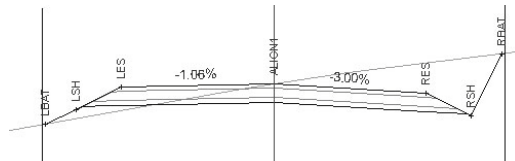


Figure 18-24. Display in the Cross Section Pane

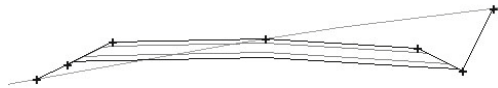



Figure 18-25. Alternative Display – Large Point Marks but with No Labels

To control the display of data in the CrossSection pane, do the following:

1. Click **Settings** ► **CrossSection** or click the **Hot Button**  at the bottom of the dialog box.

The *Road Display Settings* dialog box appears (Figure 18-26).

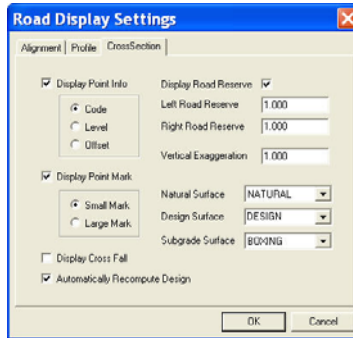


Figure 18-26. Road Display Settings – Cross Section Tab

2. Make sure the *CrossSection* tab is opened, and modify the properties as required in Table 18-8.
3. Click **OK** to close the dialog box and refresh the display on the screen.

Table 18-8. Cross Section Field Descriptions

Field Name	Field Description
Display Point Info	Select either Code, Level, or Offset for the points on the current surface.
Display Point Mark	Displays the point marks on the current surface. The size of the mark may be set to small or large.
Display Cross Fall	Displays the cross fall on each leg of the cross section of the current surface. Cross falls are hidden if the mapping scale is set so the data overlaps.
Automatically Recompute Design	Automatically recomputes (updates) the design in the drawing area after you have finished entering data.
Display Road Reserve	Displays vertical yellow lines on the cross section where the two offsets in the next two fields represent the left and right road reserve or a know limitation of the design corridor. The center line is marked with a vertical green line. These lines are a useful guide to mark specified offsets.
Left Road Reserve	Enter a value for the left road reserve.
Right Road Reserve	Enter a value for the right road reserve.

Table 18-8. Cross Section Field Descriptions (Continued)

Field Name	Field Description
Vertical Exaggeration	The default value is 1.000. Increase to change the vertical scale as a ratio to the horizontal scale in the on-screen display. A value of 2.000 sets the vertical scale at twice the horizontal scale.
Natural Surface	The natural surface displayed in the Cross Section and Profile panes. This surface is set when cross sections are extracted from the Survey View. The surface set in this field is the current natural surface. The road design is computed relative to the current natural surface with slopes intersecting with this surface.
Design Surface	This is displayed in the Cross Section and Profile panes. The surface set in this field is the current design surface. You can have more than one design surface.
Subgrade Surface	Multiple subgrade surfaces may be computed, representing different materials of varying thicknesses. The surface set in this field is the current subgrade surface.

Alignment Menu

This section describes the different Roads View menu commands in the **Alignment** menu located on the main menu bar (Figure 18-27 on page 18-28).

The Alignment menu is available when the Alignment pane is the current pane of the Roads View. To make the Alignment pane active, click in this pane with the mouse. The menu name will now rotate to the “Alignment” menu.

In the Alignment menu, you can add, edit, and delete strings used for stringline design or control the string data created from the design template and batters. Final design road data may be selected and transferred to the Survey View.

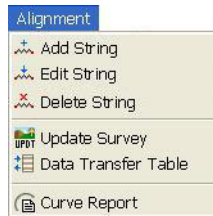


Figure 18-27. The Alignment Menu – Roads View

In This Section:

- “Add String”
- “Edit a String” on page 18-30
- “Delete String” on page 18-31
- “Update Survey” on page 18-31
- “Data Transfer Table” on page 18-31
- “Curve Report” on page 18-34

Add String

Use the Add String command to add a new string to the design based on a fixed offset, coded offset and extension, or another alignment or string in the Survey. Check mark the *Active* box to attach a template to enable the string to accept various templates (Figure 18-28 on page 18-30).



Be sure your cursor is in the Alignment pane, otherwise, the Alignment Menu option is not available.

To use the Add String command, do the following:

1. Click **Alignment ▶ Add String**. The *Add String* dialog box appears (Figure 18-28 on page 18-30).
2. On the *Add String* dialog box, set the following parameters.
 - Name – enter a name for the string, with either an alpha numeric number or a combination of numbers and alphabets.

- Active – check mark this box if you want to add a template to make the string an active design string and to enable the string to accept various templates.
 - Fixed Offset – the distance from the alignment, right or left, that the new string where the new string is created.
 - Coded Offset – an added string defined by a specific code.
 - Extension – offset distance from the specified code, either Coded Offset, Alignment, or String.
 - Alignment – select an Alignment name from the drop-down entry box to be added by an existing alignment.
 - String – select a String type from the drop-down entry box.
 - String by Intersecting Grade – select to create a string at the intersection of two grades. A right and left string reference is specified and then the user enters the grade from each of the two references. A string will be created where the two grades intersect.
 - Start Chainage – enter a value for the start chainage of the design string.
 - End Chainage – enter a value for the end chainage of the design string.
 - Color – select the desired color from the drop-down entry box for the design string.
 - Plotting Label – enter a label to be used in the profile plot.
 - Plot Label: check mark this box to display the label you want to plot.
 - Plot Design Details: check mark this box to plot design levels.
3. Click **OK** to save and exit to the main screen.
 4. Click **Cancel** to quit and return to the main screen.

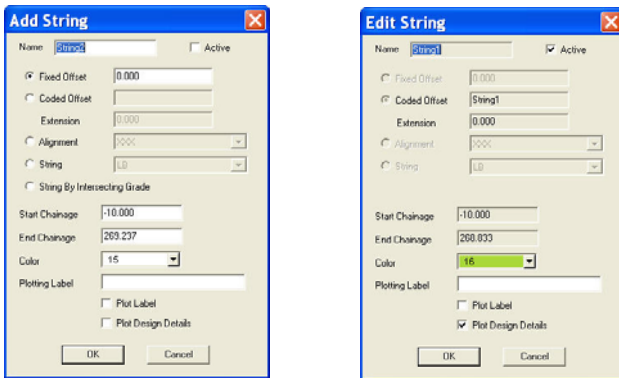


Figure 18-28. The Add String/Edit String Dialog Box

Edit a String To use the Edit String command, click **Alignment ► Edit String** (the *Edit String* dialog box displays), to edit the parameters found in the *Add String* dialog box (Figure 18-28). You can change the color, make a string active and alter the plotting label of a selected string.



Parameters for the Edit String command are the same as the Add String command.



The Edit String dialog box will appear when you double-click a string.

Delete String

Select the string of text you want to delete and click **Alignment ▶ Delete String**. The string is deleted from the design (Figure 18-29).



Figure 18-29. Delete String Dialog Box

If a design string is selected and deleted, templates that were attached to that string will disappear from the design.

Update Survey

The Update Survey command updates the Survey View with the surface strings and a surface models, as defined in the Data Transfer Table.

Click **Alignment ▶ Update Survey** to update the survey module view with selected design strings as selected in the Data Transfer table. Before this option can be executed, the data transfer table must be set up. If Update Survey is selected before the data transfer table is set up, a table will be provided for you.



Before running this option, complete the design and set up the data transfer table.

Data Transfer Table

Use the Data Transfer Table command to set up parameters for transferring road data back into the survey module. The Update Survey option must be used first.

- To use the Data Transfer Table command, click **Alignment ► Data Transfer Table**. The *Transfer Data to Survey* dialog box appears (Figure 18-30).

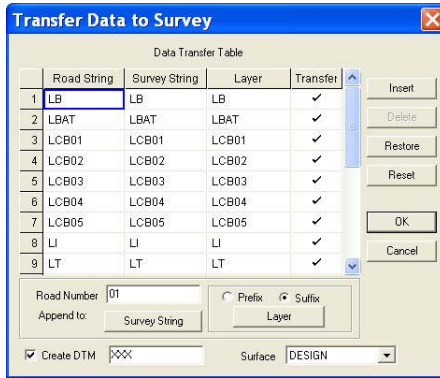


Figure 18-30. Transfer Data to Survey Dialog Box

Table 18-9 describes the parameters to set on the *Transfer Data to Survey* dialog box:

Table 18-9. Transfer Data to Survey Parameter Description

Field	Descriptions
Road String	The name of each string in the road design. These strings comprise the reference alignment, the strings created for each labeled template leg, which has its “Plot” field check marked, and certain standard strings named by the software. NOTE: The standard string names include those used for batter intercepts, table drains, verges, and compound batter sections.
Survey String	The name used for this design string when created in the Survey View. This is set to the same name as the Road String, but may be modified to a different name.
Layer	The layer created to hold this string and all its node points in Survey View. This is set to the same name as the Road String, but may be modified.
Transfer	Check mark this field to transfer a string to the Survey View with the Survey Update option. If unchecked, the string will not be written into the Survey View.

Table 18-9. Transfer Data to Survey Parameter Description (Continued)

Field	Descriptions
Road Number	If a project has more than one road, enter a different number for each road in this field.
Survey String	Press this button to Append (add) the road number to the Survey String column.
Layer	Press this button to prefix or suffix the Road Number name either before or after the parameters in the Layer column. Use this option when different layers are needed for each road in the Survey View.
Create DTM	Check mark this box and enter a name to create a digital terrain model for the selected surface defined in the Surface field.
Surface	Select the surface to be transferred to the Survey View from the drop-down entry box. This will normally be the Design surface and the default string names will display for the Design surface. NOTE: If the surface name is changed, the Road Strings, Survey Strings, and the Layer names are reset to match the relevant Road Strings for that surface.
Insert	Insert a new row.
Delete	Delete a highlighted row.
Restore	Restore a highlighted row.
Reset	Reset to default parameters.
OK	Press to save the settings, then select Update Survey option in the Alignment menu.
Cancel	Quit without saving settings and exit to the main screen.

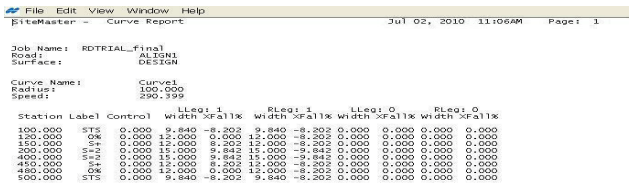


A complete string is created in the Survey View for each Road String check marked in the Data Transfer Table. Each cross section position is defined by a node point, which is coordinated with its relevant level. A surface model displays. The DTM uses standard settings but may be edited using the DTM Settings option in the Surface menu to apply one of the pre-defined Contour tables.

Curve Report

Use the Curve Report command to display a report listing the design alignment chainages through each curve with the changes in superelevation and widening applied by the active superelevation tables.

Click **Alignment ► Curve Report**. The *Curve Report* dialog box appears (Figure 18-31).



Station	Label	Control	Width	S11%	S21%	S31%	S41%
100.000	STS	0.000	9.840	-8.202	9.840	-8.202	0.000
120.000	OK	0.000	12.000	0.000	12.000	-9.202	0.000
150.000	S+1	0.000	12.000	8.202	12.000	-9.202	0.000
200.000	S+2	0.000	12.000	9.840	12.000	-9.842	0.000
400.000	S+2	0.000	12.000	9.842	12.000	-9.842	0.000
450.000	S+1	0.000	12.000	8.202	12.000	-9.202	0.000
480.000	OK	0.000	12.000	0.000	12.000	-9.202	0.000
500.000	STS	0.000	9.840	-8.202	9.840	-8.202	0.000

Figure 18-31. The MAGNET Office Curve Report

You can read the report on screen, print it out, or save it to the report (.rep) file.

Profile Menu

This section describes the different Roads View commands on the Profile menu located on the main menu bar (Figure 18-32 on page 18-35).

The Profile menu is available when the Profile pane is the current pane of the Roads View. To make the Profile pane active, click in this pane with the mouse. The menu name will then rotate to the “Profile” menu (Figure 18-32 on page 18-35).

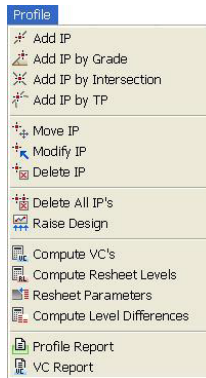


Figure 18-32. Profile Menu Options

In This Section:

- “Add IP”
- “Add IP by Grade” on page 18-37
- “Move, Modify, or Delete an IP” on page 18-39
- “Raise Design” on page 18-39
- “Compute VC’s” on page 18-40
- “Resheet Parameters” on page 18-45
- “Compute Level Differences” on page 18-47
- “Profile Report” on page 18-48
- “VC Report” on page 18-48

Add IP

The Add IP command is used to design the longitudinal profile of the road. You can insert a new IP at a nominated chainage and level, with other parameters available for curve definition. The new IP is automatically incorporated into the design.

To add an IP to the design, do the following:

1. Click **Profile ► Add IP** or click the *IP* icon on the toolbar. The Input bar in Figure 18-33 displays.



Figure 18-33. Adding an IP

A small triangle is attached to the mouse cursor.

2. Enter either the Chainage and/or Level for the new IP position or click an approximate position onscreen with the mouse. The *IP Details* dialog box appears (Figure 18-34).

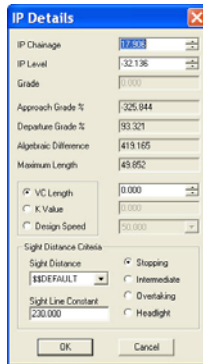


Figure 18-34. IP Details Dialog Box

Table 18-10 describes the IP Details field options.

Table 18-10. IP Details Field Descriptions

Field Type	Field Description
IP Chainage	Enter a chainage value for the mouse position.
IP Level	Enter a level value for the mouse position.
Grade	Not available.
Approach Grade %	Shows the grade of the road approaching this IP from the previous IP at the lower chainage. This value changes if the chainage and level are modified.
Departure Grade %	Shows the grade of the road departing from this IP to the next IP at the higher chainage. This value changes if the chainage and level are modified.
Algebraic Difference	Shows the change in grade, which is calculated as: <ul style="list-style-type: none"> • Departure grade (%) minus Approach grade (%)

Table 18-10. IP Details Field Descriptions (Continued)

Field Type	Field Description
Maximum Length	Shows the maximum curve length available to fit a vertical curve at this IP, which is twice the distance from this IP to the nearest IP or VC end point.
VC Length	If VC length is selected, enter the length of the parabolic vertical curve if the radio button is set to the Vertical Curve length.
K Value	If K Value is selected, enter the appropriate K Value. This is the value required for a 1% change in grade. The VC Length is computed from this value: <ul style="list-style-type: none"> • $K=L/A$, where L=Curve and A=Algebraic Difference in grades
Design Speed	If Design Speed is selected, select the speed from the drop-down list to compute the VC length from the Sight Distance Criteria parameters below.
Sight Distance Criteria: Used to compute suitable curve lengths when Design Speed is selected.	
Sight Distance	Select a Sight Distance table from the drop-down list. <ul style="list-style-type: none"> • These tables can be found in the Roads Library and relate the design speed to suitable sight distances allowed for in local standards such as Austrroads. • Select either Stopping, Intermediate, Overtaking, or Headlight distance.
Sight Line Constant	This is set from the Design Speed.
OK	Press to save the data.
Cancel	Press to quit.

Add IP by Grade

Use the Add IP by Grade command to insert a new IP at a grade from an existing IP, with other parameters available for curve definition.

To add an IP to the design by grade from an existing IP, do the following:

1. Click **Profile** ► **Add IP by Grade** or click on the *Add IP by Intersection* icon on the tool bar.

A small triangle is attached to the mouse cursor and the following prompt line displays: “*Select first IP*”.

2. Click the first IP with the mouse. This IP will have the lower chainage of the two IPs.
3. Move the mouse to display the approximate grade with a yellow construction line or enter a grade in the Grade field (Figure 18-35).



Figure 18-35. Enter a Grade for the First IP

The following prompt line displays: “Select the second IP”

4. Click the second IP with the mouse.

The following prompt line displays: “Input the grade from the second IP”

Move the mouse to display the approximate grade with a yellow construction line or enter a grade in the Grade field (Figure 18-36)

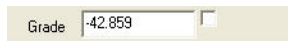


Figure 18-36. Enter a Grade for the Second IP

5. Double-click on the mouse to display the *IP Details* dialog box (Figure 18-34 on page 18-36).
6. Press **OK** to save the data or **Cancel** to quit.

The triangular symbol is displays for the new IP and the vertical profile design is adjusted.

Add IP by Intersection

The Add IP by Intersection option adds a vertical IP at the chainage and level calculated by the intersection of grades from two existing vertical IPs.

Add IP by TP

The Add IP by TP option adds an IP based upon selected TPs and the slope of the tangent segment. A best fit vertical curve will be calculated for the created IP.

Move, Modify, or Delete an IP To modify, move, or delete an IP, do the following:

1. Click **Profile ▶ Modify IP, Profile ▶ Move IP, or Profile ▶ Delete IP**

One of the following prompt lines will appear, depending on your selection:

- Select IP to move
 - Select IP to edit
 - Select IP to delete
2. Use the mouse to click on the appropriate IP to modify, move, or delete.
 3. To delete all IPs, click **Profile ▶ Delete All IPs**. A prompt displays to confirm that you want to delete all IPs.

Raise Design

The Raise Design option allows you to either raise or lower the vertical design IP levels on the current string by a nominated value. The levels can be modified for the whole length of the road or for only a section of the road.

To raise or lower the design, do the following:

1. Click **Profile ▶ Raise Design**. The *Raise IP's* dialog box appears (Figure 18-37).

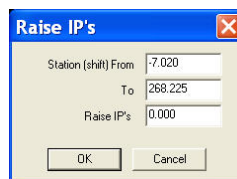


Figure 18-37. Raise a Design IP

2. Set the following parameters on the *Raise IP's* dialog box:
 - Station (shift) From – This is the first offset chainage on the current string. Enter another value to define a section of the road design.
 - To – This is the last offset chainage of the current string. Enter another value to define a section of the road design.
3. Press **OK** to cancel or **Cancel** to quit.

Compute VC's

The Compute VC's command allows you to create a longitudinal design for a defined and selected string computed from levels on the natural surface or from the design levels of another string. The option creates a VIP (vertical intersection point) at each cross section and you can then edit this profile design if necessary.



This option is the basis of stringline design. It is normally used in road reconstruction work to calculate levels for the different strings involved in the design, such as the addition of curb and channels to an existing road, or to widen a lane of a road using the existing pavement grading.



The computed levels are based on existing levels for surveyed strings and design alignments using nominated grades and offsets from the existing information.

To compute the new VIPs on a string, do the following:

1. Click **Profile ► Compute VC's**. The *Compute VC* dialog box appears (Figure 18-38).

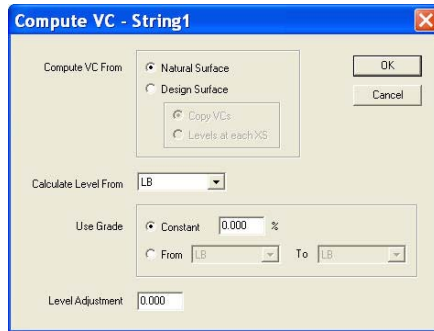


Figure 18-38. Compute VCs

Table 18-11 describes the Compute VC menu options.

Table 18-11. Compute VC's Field Descriptions

Field	Description
Compute VC From	
Natural Surface	<p>If selected, the levels will be computed from the natural surface levels on each cross section at the position defined by the string selected in the Calculate Level From field below.</p> <p>New IPs are calculated at each cross section with a curve length of 0.00m.</p>
Design Surface	<p>If selected, the levels will be computed from the design levels as specified by either Copy VCs or Levels at each XS.</p> <ul style="list-style-type: none"> • Copy VCs: New IPs are created only at the chainages of the existing VCs of the design string. The IPs on the string will be at the offset chainage along that string and the curve lengths are set at the same length as those on the design string. • Level at each XS: New IPs are calculated at each cross section with a curve length of 0.00m.

Table 18-11. Compute VC's Field Descriptions (Continued)

Field	Description
Calculate Level From	<p>Select a string from the drop-down list. The levels on the new IPs are computed from the levels on this string.</p> <p>NOTE: The string may be an existing string defined from an alignment, coding from surveyed data, or from a string already computed in an intermediate design.</p>
Use Grade	<p>The design levels being computed are based on the levels of the string selected in the Calculate Level From field, using the grade selected in this section.</p>
Constant	<p>If selected, enter a constant grade or cross fall from the level on the string defined in the Calculate Level From field to the current string. The grade may be positive or negative.</p>
From/To	<p>Select a string from the drop-down list to define the grade from existing data between one string and another. This is computed separately for each cross section.</p> <p>NOTE: This is ideal for computing the existing cross fall when the road is being widened, existing grades between the center line and the edge of the road, or two other suitable strings.</p>
Level Adjustment	<p>Enter a level that is added to the computed IP at each cross section. Use this if the existing surface is being overlaid. If the existing surface is part of the design surface, leave this field set 0.00.</p>

2. Press **OK** to proceed with the computation.
3. Press **Cancel** to quit.



The computed VC levels display in a new window in Report View. The report can be read onscreen, printed out, or saved to a report (.rep) file. When the report is closed, the triangular IP symbols display at each chainage on the longitudinal profile.



If IPs already exist for the selected string, a warning message displays. These may be overwritten by the new computation.



- If you want a simple overlay on the existing surface, you can use the Compute VC option for the strings defining the surface. Calculate the level for each string from its own existing value (Constant grade of 0%) and add the overlay in the Level Adjustment field. MAGNET Office joins the computed levels of the strings to create the new pavement for each cross section. Templates and side slopes can be allocated to the left and right of the outer strings.
- If no templates are defined on strings that have IP levels, the cross section surface is defined from left to right by joining the IP levels on each string.

Compute Resheet Levels

The Compute Resheet Levels command allows you to compute a design profile that gives a minimum depth cover over the existing pavement. MAGNET Office calculates a series of VIPs (vertical intersection points) with zero vertical curve length for each chainage of the natural surface. You can modify the vertical curve design to suit, but bear in mind that the IP level computed at any cross section is the minimum overlay. For example, you will often need to define curve lengths at some VIPs.



This technique may be known to some designers as Road Resheeting, Shape Correction, or Overlay Design.

Before this option can be run, you must set up and apply Templates and Superelevation Tables on the Design menu to define the finished shape required for the design. Other parameters for the computation are defined in the Resheet Parameters table.

The resheet levels are computed from the design cross falls at each cross section. They are applied to any point on the natural surface at

or within the string extents specified in the Resheet Parameters table. The shape of the finished cross section, complete with superelevation, is laid on the current surface to find the critical high point. The minimum overlay depth is added to the high point and the resulting level at the center line is adopted for the design IP level.



Once the IP levels have been computed, the final design is completed using the other tools in Roads View.

To compute new design VIPs on a string using the Resheet options, do the following:

1. Click **Profile** ► **Compute Resheet Levels**. A confirmation screen appears to warn you that existing IPs will be overwritten.
2. Press **OK**. The **Compute Resheet Report** dialog box appears (Figure 18-39).

Compute Resheet Report Jul 02, 2012 11:06AM Page: 1

Job: C:\Data\ROAD\0567.ccx

Chainage	Shift Offset	Left Limit	Left xFall	Right Limit	Right xFall	Min Depth	Natural Level	Design Level
-10.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	48.69	48.69
0.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	49.33	49.33
10.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	49.69	49.69
20.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	50.05	50.05
30.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	50.40	50.40
40.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	50.76	50.76
42.416	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	50.85	50.85
45.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	51.12	51.12
46.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	51.47	51.47
48.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	51.83	51.83
50.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	52.18	52.18
60.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	52.41	52.41
100.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	52.41	52.41
110.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	52.17	52.17
120.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	51.70	51.70
127.482	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	51.19	51.19
130.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	50.99	50.99
140.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	50.05	50.05
150.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	49.01	49.01
158.377	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	48.13	48.13
160.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	47.96	47.96
170.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	46.99	46.99
180.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	46.43	46.43
190.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	46.31	46.31
191.141	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	46.31	46.31
200.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	46.65	46.65
220.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	47.46	47.46
230.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	48.72	48.72
230.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	50.18	50.18
240.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	52.09	52.09
250.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	51.82	51.82
260.000	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	55.54	55.54
269.337	0.001000000000.00	-1.00	-1000000000.00	-1.00	0.00	0.00	55.99	55.99

Figure 18-39. Compute Resheet Report Dialog Box

When the report is closed, the triangular IP symbols display at each chainage on the longitudinal profile.

If IPS already exist for the selected string, a warning message displays. These may be overwritten by the new computation.



If a simple overlay is to be laid over the existing surface, you may use the stringline options instead. For example, the Compute VC option is run for each of three strings at the center, left, and right of the existing pavement. Calculate the level for each string from its own existing value (Constant grade of 0%) and add the overlay in the **Level Adjustment** field. MAGNET Office joins together the computed levels of the strings to create the new pavement for each cross section. Templates and batters may then be allocated to the outer strings.

Resheet Parameters

Road overlays or resheeting requires the user to either define parameters for computing a minimum overlay or resheet to handle jobs where an existing road is being resurfaced. This process may be a simple overlay of the existing surface or incorporate a certain amount of shape correction to produce a road with suitable superelevation through existing curves. The process enables the user to compute a longitudinal design that allows a specified cross section template to be overlaid on the existing road surface at a minimum depth over the highest point of that surface on each cross section. A Vertical Intersection Point is created at each cross section with a zero length curve. The levels of these points represent the minimum levels to achieve the overlay but in many cases the user will have to fine tune this design to produce suitable grading along the vertical curve.

To set the parameters for computing overlays, do the following:

1. Click **Profile ▶ Resheet Parameters**. The *Resheet Parameters* table appears (Figure 18-40).



Figure 18-40. Resheeting Parameters

Table 18-12 describes the Resheet Parameters options.

Table 18-12. Resheet Parameters Field Descriptions

Field	Description
Chainage	Enter the chainage for the resheet parameters on the first row of the table. If the minimum depth, left string, and right string are the same for the full length of the road, no further rows of data are entered.
Min. Depth	Enter the minimum depth of the resheet or overlay in meters.
Left String	Enter the string that defines the left edge of the computation of the overlay from the drop-down list.
Right String	Enter the string that defines the right edge of the computation of the overlay from the drop-down list.
Insert	Press to insert a desired parameter.
Delete	Press to delete a selected parameter.
Restore	Press to restore a deleted parameter.
OK	Press to save and return to the previous screen.
Cancel	Press to quit the routine.



These parameters are read when the **Profile ▶ Compute Resheet Levels** command is used.

Compute Level Differences

The Compute Level Differences command creates a report listing the offsets between two nominated design strings, with the level differences between two nominated surfaces. A minimum depth may be set so that all level differences that are less than this minimum are flagged on the report. This report is useful for checking that sufficient overlay has been applied to a design surface when completing resheet or overlay projects.

To create a report showing level differences, do the following:

1. Click **Profile** ► **Compute Level Differences**. The *Compute Level Differences* dialog box appears (Figure 18-41).

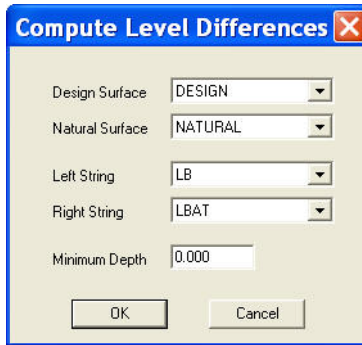


Figure 18-41. Compute Level Differences

2. On the *Compute Level Differences* dialog box, select or enter the following parameters, then press **OK** to save or **Cancel** to quit.
 - Design Surface – Select a design surface from the drop-down list.
 - Natural Surface – Select a natural surface from the drop-down list.
 - Left String – Select the string that defines the left edge of the offsets in the range from the drop-down list.
 - Right String – Select the string that defines the right edge of the offsets in the range from the drop-down list.

- Minimum Depth – Enter a minimum depth. Level differences less than this minimum depth will be flagged on the report.



The level differences for each chainage in the road display in a new report window. The report may be read onscreen, printed out, or saved to a report (.rep) file.

Profile Report

To create a string chainages report, click **Profile ▶ Profile Report**. The level differences for each chainage in the road display in a new report window (Figure 18-42).

You can read the report onscreen, print it out, or save to a report (.rep) file.

String Chainages Report Jul 02, 2012 11:06AM Page: 1

Road Name: XXX
String Name: String1

Chainage	ShiftChainage	Offset	Level
-10,000	-10,000	0.000	48.691
0,000	0,000	0.000	49.334
10,000	10,000	0.000	49.691
20,000	20,000	0.000	50.047
30,000	30,000	-0.000	50.404
40,000	40,000	0.000	50.760
42,416	42,416	0.000	50.847
50,000	49,996	0.000	51.116
60,000	59,983	-0.000	51.473
70,000	69,972	0.000	51.829
80,000	79,960	0.000	52.185
90,000	89,949	-0.000	52.404
100,000	99,937	-0.000	52.160
110,000	109,926	0.000	51.691
120,000	119,914	0.000	51.184
127,482	127,482	0.000	50.987
130,000	130,000	0.000	50.542
140,000	139,989	0.000	48.996
150,000	149,978	0.000	48.120
158,177	158,177	0.000	47.940
160,000	159,966	0.000	46.974
170,000	169,955	0.000	46.410
180,000	179,944	0.000	46.309
191,163	191,160	0.000	46.334
200,000	199,929	0.000	46.668
210,000	209,898	0.000	47.488
220,000	219,867	0.000	48.770
230,000	229,836	0.000	50.443
240,000	239,805	0.000	52.164
250,000	249,774	0.000	53.885
260,000	259,743	0.000	55.406
269,217	269,833	0.000	56.012

Figure 18-42. Create a Profile Report

VC Report

The VC Report command lists the details of the Ips defined for the vertical profile design of the road on the current string.

1. To list the Ips, select the desired string on the Property Toolbar, then click **Profile ▶ VC Report**.
2. The **VC Details** dialog box appears (Figure 18-43 on page 18-49).

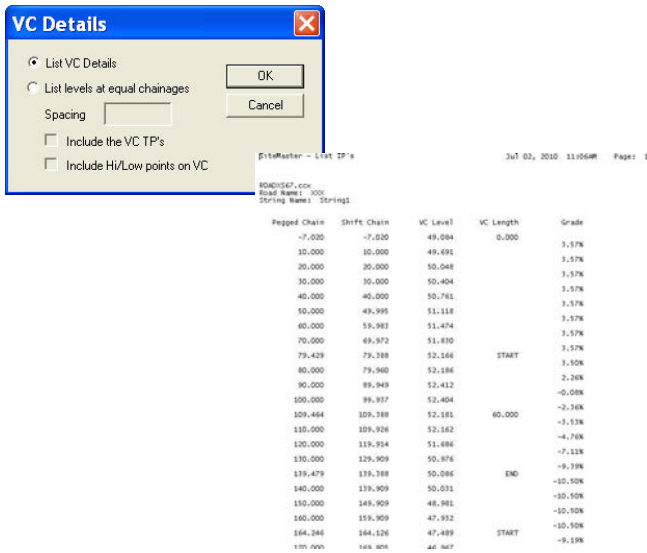


Figure 18-43. Create a VC Report

- On the *VC Details* dialog box, select or enter the following parameters:
 - List VC Details – select this option to list the VC details in a simple report as shown in Figure 18-43.
 - List levels at equal chainages – select this option to include further items in the list, and to set a designated spacing for the details on the VC.
 - Include the VC TP's: check mark this box to include the details of the tangent points at each end of all parabolic curves.
 - Include Hi/Low points on VC: check mark this box to include the details at the high and low points on the design profile.

After the desired format has been selected, click **OK** to produce the VC Report. The report may be read onscreen, printed out, or saved to a report (.rep) file.

- Click **Cancel** to quit the routine.

CrossSection Menu

This section describes the different CrossSection menu commands located on the main menu bar (Figure 18-44).



Figure 18-44. Cross Editor Design Menu

In This Section:

- “Add Surface” on page 18-51
- “Subgrade Sequence” on page 18-53
- “Delete Surface” on page 18-54
- “Add Point” on page 18-55
- “Raise Point” on page 18-56
- “Adjust Side Slope” on page 18-57
- “Intersect Side Slope” on page 18-58
- “Strip Depth” on page 18-59
- “Omit Stations” on page 18-61
- “Volume Factors” on page 18-60
- “Compute Volumes” on page 18-62
- “Updating Changes in Survey View” on page 18-64
- “Cross Sections Report” on page 18-65
- “Construction Report” on page 18-65

Add Surface

Use the Add Surface command to add another surface to the road design. This surface is usually a subgrade surface but can be an alternate design surface. The first subgrade surface defaults to BOXING, which you can rename unless a design has already been computed.



The order of the subgrade surfaces is initially the order in which they are added from the top subgrade surface down; however, you can change the order using the **CrossSection ▶ Subgrade Sequence** command.

To add another surface to the road design, do the following:

1. Click **CrossSection ▶ Add Surface**.

The *Add Surface* dialog box appears (Figure 18-45).

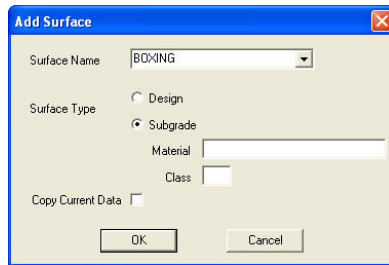


Figure 18-45. Add Surface Dialog Box

2. Define the surface as required.

For field descriptions, see Table 18-13 on page 18-52.

3. When you have finished, click **OK** to create the new surface or **Cancel** to quit.

The surface becomes the current Design or Subgrade surface in the relevant field on the **Cross Sections** tab of the *Hot Button*  dialog box.

4. To set the new surface as the current surface in the Cross Section pane, select it from the surface drop-down menu on the Properties toolbar (Figure 18-46 on page 18-52).

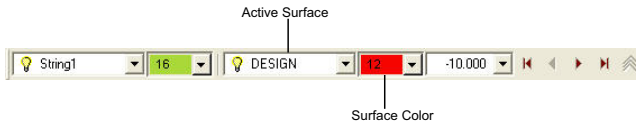


Figure 18-46. Properties Toolbar – Surface and Surface Color Menus

- To set the color of the new surface, select on in the color box adjacent to the surface box on the **Properties** toolbar.

Table 18-13. Add Surface Field Descriptions



Field	Description
Surface Name	This is the surface name. The default is BOXING. Enter a name that suits the definition of the surface, so you can identify it. You can choose surfaces, such as Basecourse, Subbase, and subgrade for a road with three subgrade surfaces or use the material name if preferred.
Surface Type	<ul style="list-style-type: none"> Design: Select this option when you want the new surface to be the design surface. This surface is listed as the current surface in the Design Surface drop-down menu on the Cross Section tab of the Hot Button  dialog box. Subgrade: Select this option when you want the new surface to be the subgrade surface. This surface is listed as the current surface in the Design Surface drop-down menu on the Cross Section tab of the Hot Button  dialog box.
Material	If the surface is a subgrade, then enter the material used. This is optional, but the material is listed on the subgrade volumes report. This field is unavailable (grayed out) for a new design surface.
Class	If the surface is a subgrade, then enter a character or characters to label and allocate the subgrade to a class. This is optional, but the class may be used to categorize the subgrade on the volumes report. This field is unavailable (grayed out) for a new design surface.

Table 18-13. Add Surface Field Descriptions (Continued)


Field	Description
Copy Current Data	Select this check box to copy the data allocated to the current design or subgrade surface to the new surface. This populates certain settings on the Design menu for the selected surface.

Add a Trial Design To add a trial design surface, do the following:

1. click **CrossSection ▶ Add Surface**. The *Add Surface* dialog box appears (Figure 18-45 on page 18-51).
2. Enter a new name in the *Surface Name* field and check mark the *Copy Current Data* box.

Once created, set the *Properties* toolbar “Surfaces” field to the new design surface, i.e., BOXING.

Additional Information

- Multiple design surfaces enables you to change to an alternative surface for plotting or volumes. You can set the current surface on the *Cross Sections* tab on the *Hot Button*  dialog box.
- You can extract cross sections as either design or natural surfaces. This allows you to build multiple surveyed surfaces doing construction or mining applications. These multiple surfaces may be used in plotting or volume calculations. If the surface will be a design surface, you can add the surface in the Cross Section Editor and then use this name for the surface when extracting the cross sections in the Survey View.

Subgrade Sequence

Use the Subgrade Sequence command to reorder subgrade surfaces. The sequence is originally defined in the order that the subgrade surfaces are created with the first surface being the top subgrade surface. The sequence should be set correctly so the appropriate surfaces intersect each other.

To reorder subgrade surfaces, do the following:

1. Click **Cross Section ▶ Subgrade Sequence**.

The *Subgrade Sequence* dialog box appears (Figure 18-47), displaying the surfaces in order as shown in the **Subgrade Sequence** list.

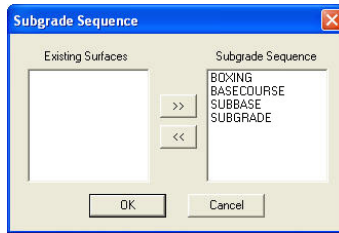


Figure 18-47. Subgrade Sequence dialog box

2. To change the order, select a surface and click the left-facing double arrows **<<** icon to move it into the **Existing Surfaces** list. It may be necessary to move all surface names to the **Existing Surfaces** list.
3. Select each surface in the **Existing Surfaces** list, starting with the surface you want to appear first in the **Subgrade Sequence** list, and click the right-facing double arrows **>>** icon to move each surface over to the sequence list.
4. If you do not want a surface to appear in the **Subgrade Sequence**, then move it to the **Existing Surface** list if you have not already moved it and leave it there.

Additional Information The volume of the first subgrade below the design surface is shown as the Pavement volume on the volumes report. The volumes of the other subgrade surfaces are combined in the Subgrade Volume; however, the volumes of each subgrade may all be listed separately by selecting the *Multiple Subgrade Volumes* option in the **Cross Section ▶ Compute Volumes** command.

Delete Surface

Use the Delete Surface command to delete any or all of the surfaces that are defined in the road dataset. You cannot delete the current natural and design surfaces.

To delete a surface defined in the road dataset, do the following:

1. Click **Cross Section ▶ Delete Surface**.

The *Delete Surface* dialog box appears (Figure 18-48).



Figure 18-48. Delete Surfaces dialog box

2. Select a surface, and then click **Delete**.
The surface is removed from the list.
3. When you have finished, click **OK** to confirm the deletion(s) or click **Cancel** to quit the command and restore the surface to the list.

If the design or natural surface is selected, then a warning message appears. You can remove the default BOXING surface if it is not being used in the design and there are other subgrade surfaces.

Add Point

Use the Add Point command to add a point to a single cross section to modify the current surface of the section displayed in the Cross Section pane.

To use the Add Point command, do the following:

1. Click **Cross Section ▶ Add Point**.

The input bar appears (Figure 18-49).



Figure 18-49. Add Point Input Bar

2. Select a point position with the mouse in the Cross Section pane or enter the data in the fields on the input bar.
3. Press **Enter** or click with the mouse to confirm the point.

The Cross Section pane updates automatically with the new point inserted between the end points (Figure 18-50).

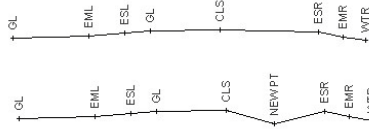


Figure 18-50. The New Point (NEWPT) is Inserted Into the Cross Section

Raise Point

Use the Raise Point command to raise or lower by a fixed amount the level on a section of the current cross section. This section is defined by an offset range. Alternatively, the height change may be applied to a point or points with a nominated code.

To use the Raise Point command, do the following:

1. Define the current surface on the **Properties** toolbar, and select the required cross section in the Cross Section pane (Figure 18-51).

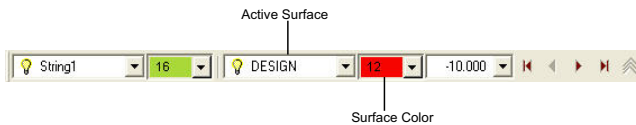


Figure 18-51. Properties Toolbar

2. Click **Cross Section ▶ Raise Point**.

The *Add Height to Xsect Points* dialog box appears (Figure 18-52).

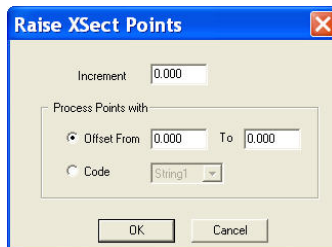


Figure 18-52. Add Height to Xsect Points

3. In the **Increment** field, enter a positive value to add to the height of the existing levels. Enter a negative value to lower the levels.
4. Select one of the following options:
 - **Offset From:** The offset extents of the cross section are displayed. Enter the offset range. The height is modified for all points within this offset range.
 - **Code:** Select an existing code. The height is modified for all points in this cross section with this code.
5. Click **OK** to change the levels.

The levels are updated and the cross section is redrawn.

Adjust Side Slope

Use the Adjust Side Slope command to adjust the slope of a single existing side slope in the current cross section by specifying an offset for the side slope intercept or a side slope.

To use the Adjust Side Slope command, do the following:

1. Choose the design surface on the **Properties** toolbar, and then select the required cross section in the Cross Section pane (Figure 18-53).

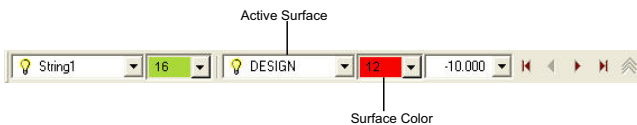


Figure 18-53. Properties Toolbar

2. Click **Cross Section ▶ Adjust Side Slope** to display the input bar.
3. Do one of the following:
 - Select an existing side slope point on the design surface with the mouse (Figure 18-54 on page 18-58).
 - Enter a new side slope offset in the input bar.
 - Select the check box to the right of the **Slope** field to enter a slope value.

The existing data in the fields change as the pointer moves.

- If you entered data into the input bar, then press **Enter** to confirm the data.
- Make each subgrade the current surface in turn and adjust the side slope point to match the design point.

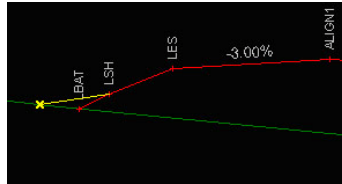


Figure 18-54. Left Side Slope Point Selected and Moved with the Pointer

- To remove a point from the cross section, select it and press **Delete**.

The point is removed, and the cross section is reformed.

Intersect Side Slope

Use the Intersect Side Slope command to add individual batters to a single cross section where no batter exists. A hinge point for the side slope is selected and a side slope or offset is entered.

To use the Intersect Side Slope command, do the following:

- Choose the design surface on the **Properties** toolbar, and then select the required cross section in the CrossSection pane (Figure 18-55).

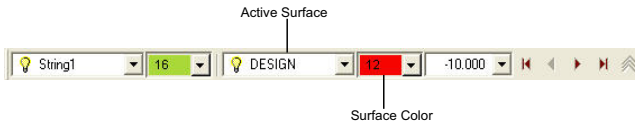


Figure 18-55. Properties Toolbar

- Click **Cross Section ▶ Intersect Side Slope** to display the input bar.
- Select the batter hinge point on the design surface with the mouse (Figure 18-56 on page 18-59).

This point is at the edge of a template from which a batter line will intersect the natural surface. A batter line is highlighted in yellow with the batter point moving along the natural surface.

- Specify the new intersection point of the batter using an offset from the center line or a batter slope (Figure 18-56).

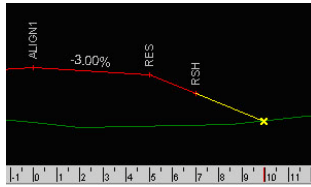


Figure 18-56. New Right Batter Point Created and Moved with the Pointer

- To remove a point from the cross section, select it and press **Delete**.

The point is removed, and the cross section is reformed.

Strip Depth

Use the Strip Depth command to specify the depth of stripping topsoil from the natural surface. This is usually stripped from the left side slope offset to the right side slope offset. The strip volume is itemized in the volume report. The stripping volume is applied to pavement volumes in the volumes report.

To use the Strip Depth command, do the following:

- Click **Cross Section ▶ Strip Depth**.

The *Strip Depth* table appears (Figure 18-57).

Strip Depth					
	Chainage	Depth	Left String	Right String	
1	0.000	0.150	LBAT	RBAT	Insert
2					Delete
3					Restore
4					
5					
6					
7					OK
8					Cancel
9					

Figure 18-57. Strip Depth

- In the first row of the **Chainage** column, enter the chainage where the stripping starts. If the stripping depth changes, specify the start chainage of the new depth on the next row.

The new strip depth is applied from that chainage. There is no interpolation between the chainages on any two rows of data. If the left and right string extents change, set up a new row of data with the start chainage for the new extents.

3. In the **Depth** column, enter the stripping depth in meters.



If a section of the road has no stripping, then specify the start chainage and a zero stripping depth with the left and right side slope strings. Do not leave the **Depth** field empty. Use this method to stop stripping from being computed for a section of road.

4. In the **Left String** column, click to display the drop-down menu and select the string that is the left edge of the stripping. This is usually the **LBAT** string. You may need to create a new string, especially if the edge of the stripping is a natural surface feature.
5. In the **Right String** column, click to display the drop-down menu and select the string that is the right edge of the stripping. This is usually the **RBAT** string. You may need to create a new string, especially if the edge of the stripping is a natural surface feature.
6. Enter data on as many rows as required to define the stripping, and then click **OK**.

The stripping volume is itemized in the volumes report when the **Cross Section ▶ Compute Volumes** command is used.

Volume Factors

Use the Volume Factors command to specify the bulking and compaction factors you want to apply to the cross section volumes. When material is removed, it generally occupies more space, and the fill is compacted to occupy less space. You can use these factors to compute the volumes resulting from a cut and required for a fill and include the effects of bulking and compaction.

To apply Volume Factors, do the following:

1. Click **Cross Section ▶ Volume Factors**.

The *Volume Factors* table appears (Figure 18-58).

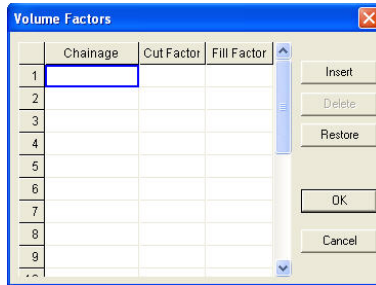


Figure 18-58. Volume Factors Table

2. Enter the required values by chainage, and then click **OK** to save.

Omit Stations

Use the Omit Stations command to omit a chainage or a range of chainages from the plotted cross sections and also from the volumes report.

To use the Omit Stations command, do the following:

1. Click **Cross Section ▶ Omit Stations**.
The *Omit Chainage* table appears (Figure 18-59).

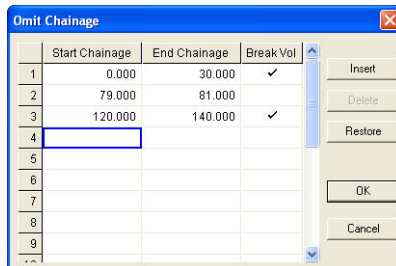


Figure 18-59. Strip Depth

2. In the first row of the **Start Chainage** column, enter the chainage where the omission starts. If one chainage is omitted from the cross section plot, the start chainage should be set at a value before the chainage being omitted, such as 79.00 above to omit chainage 80.00.

3. In the **End Chainage** column, enter the chainage where the omission ends. This will define a chainage range. If one chainage will be omitted from the cross section plot, then the start chainage should be set at a value before the chainage being omitted, such as 79.00 above to omit chainage 80.00.
4. In the **Break Vol** column, double-click to insert a check mark if you want to break the volumes report between these two chainages. Do not select this field if you want to omit chainages from the plot.

Compute Volumes

Use the Compute Volumes command to create a customized volumes report that you can save and print. The information displayed on the report can be defined by various settings.

To create a customized volumes report, do the following:

1. Complete the roads design, and then click **Cross Section ▶ Compute Volumes**.
The *Volumes Report* dialog box appears (Figure 18-60).

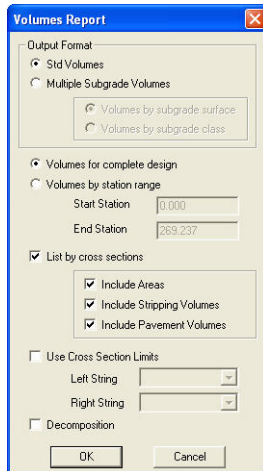


Figure 18-60. Volumes Report

2. Define the details for the volumes report.

For field descriptions, see Table 18-14 below.

3. When you have finished, click **OK** to create the report. The volumes report is displayed in a new dialog box. You can read the report on the screen, print it, or save it to a report (.rep) file.

Table 18-14. Volumes Report Menu Options

Field Name	Field Description
Output Format	Select one of the two options: Std Volumes: This produces the volumes report displaying the standard format. You can modify this format using the other settings on the dialog box. Multiple Subgrade Volumes: This produces a report listing the volumes of the various subgrade layers in the design. You can format this report in one of two ways: <ul style="list-style-type: none"> • Volumes by subgrade surface: Volumes are listed for each subgrade layer. • Volumes by subgrade class: Volumes are listed by the class allocated to the subgrade surfaces defined in the Add Surface command.
Volumes for complete design	This computes volumes for the full length of the road design.
Volumes by station range	This limits the computation of volumes between the start chainage and end chainage entered into the named fields.
List by cross sections	This lists the volumes between each cross section in the dataset with total volumes at the end of the report. Clear this check box to produce a summary of the total volumes.
Include Areas	Check mark this box to include the areas of cut and fill between adjacent cross sections.
Include Stripping Volumes	Check mark this box to include the stripping volumes between adjacent cross sections plus the total stripping volume. To include these volumes, define the required stripping parameters in the Strip Depth command.

Table 18-14. Volumes Report Menu Options (Continued)

Field Name	Field Description
Include Pavement Volumes	Check mark this box to include the pavement volumes between adjacent cross sections plus the total pavement volume. The pavement volume is the top subgrade layer between the design surface and the top subgrade surface. If only one subgrade is defined, then it will be the pavement volume.
Use Cross Section Limits	Check mark this box to limit the volume computation between the left string and right string selected in the named fields.
Decomposition	This option is not implemented.

Additional Information for Computing Volumes

- The volumes are computed between the lowest subgrade of the road and the natural surface, other volumes for the pavement (top subgrade), and all other subgrades and stripping if used. The total cut and fill volumes with the balance of material remaining are listed at the end of the report.
- To compute volumes between any two nominated surfaces, use the **Surface ▶ Cross Sections ▶ Cross Sections Volumes** command in the Survey View. Using this command, you can enter the two surfaces, and the volumes are computed from the dataset in the Roads View.
- The computation method applies the center of mass corrections to ensure accurate computations through the curves on the alignment.

Updating Changes in Survey View

Use the Update From Survey command to re-extract the cross sections from within the Roads View if any changes were made in the Survey View to the alignment of the surface models.

To use the Update from Survey command, click **Cross Sections ▶ Update from Survey**. The cross section data is re-extracted from the Survey View.

Cross Sections Report

Use the Cross Section Report command to produce a report showing a cross section listing for the current surface. The report can be saved and printed.

To produce a report showing cross section listing for the current surface, do the following:

1. Define the surface required in the report from the surface drop-down menu on the **Properties** toolbar (Figure 18-61).

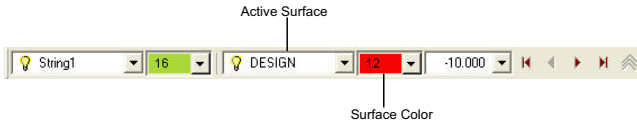


Figure 18-61. Properties Toolbar

2. Click **Cross Section** ▶ **Cross Section Report**.

The report displays in a new dialog box, listing all of the details of the cross sections for the current surface. You can read the report on the screen, print it, or save it to a report (.rep) file (Figure 18-62).

Road Name:	000				
Surface Name:	DESIGN				
Station	Offset	Easting	Northing	Elev	Code
-10.000	0.000	899.480	4968.776	48.691	String1
0.000	-6.018	904.020	4979.540	49.006	LEAT
0.000	-5.700	904.218	4979.266	49.144	LCRD1
0.000	-3.700	905.194	4977.648	49.148	LB
0.000	-3.500	905.512	4977.487	49.144	LT
0.000	-3.450	905.541	4977.466	49.104	LI
0.000	-3.000	905.806	4977.082	49.244	LB
0.000	0.000	907.149	4974.851	49.124	String1
0.000	3.000	909.371	4972.228	49.244	RB
0.000	3.450	909.587	4971.804	49.144	ET
0.000	3.700	909.924	4971.662	49.144	EB
0.000	5.700	910.920	4970.044	49.144	RCRD1
0.000	6.336	911.294	4969.530	48.708	RBAT
10.000	-6.176	912.028	4985.530	49.224	LEAT
10.000	-5.700	912.308	4985.145	49.701	LCRD1
10.000	-3.700	913.484	4981.527	49.701	LB
10.000	-3.500	913.601	4981.165	49.701	LT
10.000	-3.450	913.631	4981.125	49.551	LI
10.000	0.000	913.895	4980.961	49.601	LB
10.000	3.000	915.459	4980.574	49.691	String1
10.000	3.450	917.687	4977.743	49.551	ET
10.000	3.700	917.716	4977.703	49.701	EB
10.000	3.700	917.834	4977.541	49.701	RB
10.000	5.700	918.010	4975.923	49.701	RCRD1
10.000	6.557	919.514	4975.229	48.843	RBAT
20.000	-6.341	920.019	4989.248	49.488	LEAT
20.000	-5.700	920.397	4991.024	50.057	LCRD1
20.000	-3.700	921.571	4989.406	50.057	LB
20.000	-3.500	921.691	4989.244	50.057	LT
20.000	-3.450	921.720	4989.204	49.907	LI

Figure 18-62. List Cross Sections Report

Construction Report

Use the Construction Report command to produce a report that shows the level differences between the design or subgrade surface and the natural surface.

This report includes the center line point by default and any other point you define. Points are entered by code with the option to specify a label for each code.

To use the Construction Report command, do the following:

1. Click **Cross Section ▶ Construction Report**.

The *Construction Report Code Entry* dialog box appears (Figure 18-63).

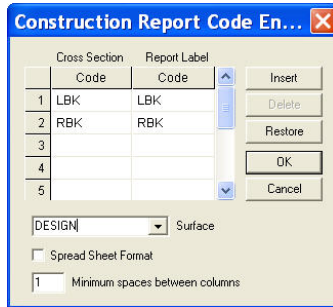


Figure 18-63. Construction Report Code Entry Table

2. In the **Cross Section** column, enter the code of the cross section points you want to include in the report.
3. In the **Printout** column, enter the label code that corresponds to the cross section point.
4. Repeat steps 2 and 3 as necessary.
5. Select a Surface type from the drop-down entry box.
6. To create a text file format, which you can load into a spreadsheet, check mark the **Spread Sheet Format** box or enter a value in the *Minimum spaces between columns* field.
7. Click **OK**. The MAGNET Office Construction Report appears (Figure 18-64).

Construction Report Jul 02, 2012 11:06AM Page: 1

Road Name: XXX
String Name: String1
Surface Name: DESIGN

Peg Chain	Shift Chain	Offset Level	Design Level	Level Diff	PT	Offset Level	Design Level	Level Diff	PT	Offset Level	Design Level	Level Diff
-----------	-------------	--------------	--------------	------------	----	--------------	--------------	------------	----	--------------	--------------	------------

Figure 18-64. Cross Section Construction Report

Design Menu

The Design menu is the main functional menu in Roads View (Figure 18-65). It contains most of the data entry tables and computational routines needed to define the standard cross section used for the final design and subgrade surfaces. The crossfall and legs of the standard cross section can be modified using superelevation tables.

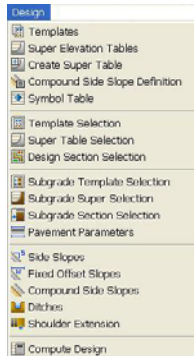


Figure 18-65. Cross Section Editor – The Design Menu

Additions or modifications are applied using the Compute Design option in the Design menu. A string must be active to apply a template and other parameters to it. The alignment string is active by default when Roads View is opened.

In This Section:

- “Templates” on page 18-68
- “Super Elevation Tables” on page 18-70
- “Create Super Table” on page 18-73
- “Compound Side Slopes” on page 18-78
- “Symbol Table” on page 18-81
- “Template Selection” on page 18-82
- “Subgrade Super Selection” on page 18-91
- “Subgrade Section Selection” on page 18-92
- “Fixed Offset Slopes” on page 18-98

- “Compound Side Slopes” on page 18-100
- “Ditches” on page 18-102
- “Shoulder Extension” on page 18-104
- “Compute Design” on page 18-105

Templates

The Templates option is used to enter details of the standard design cross section into a template that is applied at a specified station range to create the new road surface. The design surface is computed when the template is attached to the formation level of the design profile at each cross section.

To set up a template for the road, click **Design ▶ Templates**. The *Template Selection* table appears (Figure 18-66).

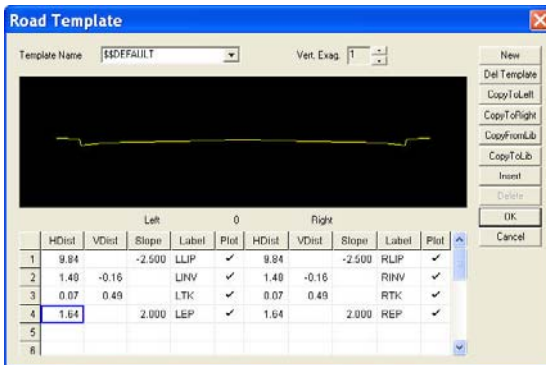


Figure 18-66. Template Selection Dialog

Table 18-15 describes the Road Template table field options.

Table 18-15. Road Template Menu Options

Field Type	Field Description
Template Name	Select a name for the template or use the default name.
Vert Exag.	Set a value to exaggeration the display of the vertical elements of the template.

Table 18-15. Road Template Menu Options (Continued)

Field Type	Field Description
Left/Right	<p>Five fields of data – Dist, VDist, Slope, Label, and Plot – apply to the legs to the left and right of the string. Enter the following data for each field:</p> <ul style="list-style-type: none"> • HDist – enter the length of the leg as the leg extends away from the string may be a negative value for some purposes, such as a subgrade template where a leg may turn back towards the centerline. • VDist – enter the height difference for this leg in meters. A positive distance raises the end of the leg and a negative distance lowers the end of the leg. NOTE: Use either VDist or Slope, but not both. • Slope – enter the crossfall of the leg as a percentage slope. A positive slope raises the end point of the leg while a negative slope lowers the end point of the leg. NOTE: If superelevation will be applied to this leg of the template, the leg must be set with a slope for its vertical element. • Label – enter a unique descriptive code for the end point of the leg. Each label defines a stringline that runs for the length of the road design through all the points labeled with the same code. NOTE: Begin each label with L for the left legs of the template or R for the right legs, so that all the left strings and all the right strings are grouped together in the drop-down list. • Plot – enter a check mark to ensure that the end point of this leg is allocated to a string and displays in the alignment pane. The string will also be available for transfer to Survey View.
New	Press to create a new template. Default names are Template1, Template2, etc.
Del Template	Press to delete the current template. You cannot delete the \$\$DEFAULT template or a template being used in the design.
CopyToLeft	Press to copy data defined in the template from the Right legs to the Left fields to create a symmetrical template. Any existing legs defined for the left side of the template are overwritten. The labels will start with L .

Table 18-15. Road Template Menu Options (Continued)

Field Type	Field Description
CopyToRight	Press to copy data defined in the template from the Left legs to the Right fields to create a symmetrical template. Any existing legs defined for the right side of the template are overwritten. The labels will start with R .
CopyFromLib	Press to display the list of templates in the library. The selected library template will be copied to the current template in the project, overwriting any existing data.
CopyToLib	Press to copy the current template to the Template Library, using its defined name. If that name already exists in the Library, you are prompted to overwrite it.
Insert	Press to create a new row in the template from the highlighted row in the template table.
Delete	Press to remove a highlighted row of data form the template. NOTE: Button becomes active only after a row is highlighted.
OK	Press to save the template.
Cancel	Press to quit the routine.



Each row of the template tables represents a leg or a segment of the template, with options to define a leg to the left and to the right of the current string. When the template definition is complete, click OK to save the template. It will be available for all roads within the project.

Super Elevation Tables

The Super Elevation Tables option allows you to enter superelevation and widening details in to individual tables so that changes can be applied to the design template at particular chainages.



Instead of typing data into these tables, use the Create Super Table option. It is much more efficient.

Each table contains a number of rows of data that map the change of the crossfall and width of particular legs defined in the template. The rate of change is dictated by the data entered on any two adjacent rows.



The information entered in this option on the Design menu is available to any road in the current project. The Super Elevation Table is applied to the template at the chainages defined in the Super Table Selection option in the Design Menu.



If a Super Elevation table is used as a standard for many projects, the details should be defined in the Super Elevation Tables option found in the Library menu because a Super Table held in the library can be copied **From** the library into a project, or a Super Table may be copied **To** the library for use in other projects. A Super Elevation table is also used to apply grade and width changes to the template used for subgrade surfaces.

To set up a super elevation table for the road, click **Design ▶ Super Elevation Tables**. The *Super Tables* table appears (Figure 18-67). The \$DEFAULT template displays, which may be used or left blank.

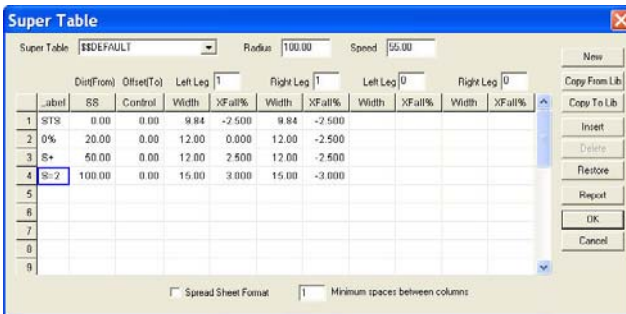


Figure 18-67. Super Table Menu

Table 18-16 describes the Super Elevation table field options.

Table 18-16. Super Elevation Table Menu Options

Field Type	Field Description
Super Table	Use the \$\$DEFAULT template or select another one from the drop-down entry box.
Radius	Enter a radius for the curve.
Speed	Enter the design speed of the table.
Left Leg/Right Leg	Enter the number relating to the left/right template leg that will be modified by the data in the two columns for Width and Crossfall beneath these fields. NOTE: Up to two template legs at each side of the template may be modified by one super table.
Label	Enter an optional label for particular points defined by each row of the super table. These will help you to check the definitions of the changes in the table.
Dist (From) (SS)	Enter the distance from the start of the superelevation changes (SS). The first row is usually set to 0 and the distances increase through the subsequent rows of data.
Offset (To) (Control)	Enter the centerline offset to allow for a shift in transitions if these are not applied in the horizontal alignment. the first row must be set to 0 if these settings are used. If this column is left blank, the offset to control is assumed to be 0.
Width	Enter the width for the relevant template leg at the distance from the SS (start of the superelevation or widening).
XFall%	Enter the cross fall as a percentage for the relevant template leg at the distance from the SS (start of the superelevation or widening).
Spread Sheet Format	Check mark this box to save the table in a spread sheet format instead of a text format.
Minimum spaces between columns	Enter the minimum number of spaces required between the columns in the file. Default is 1.
New	Press to crate a new table. Default names are Super1, Super2, etc.
CopyFromLib	Press to display the list of super tables in the library. The selected Library table is copied to the current table in the project, overwriting any existing data.

Table 18-16. Super Elevation Table Menu Options (Continued)

Field Type	Field Description
CopyToLib	Press to copy the current super table to the Super Table Library using its defined name. If the name already exist in the Library, you rae prompted to overwrite it.
Insert	Press to create a new row from the highlighted row in the table.
Delete	Press to delete a highlighted row in the table. This button will not be active until a row is highlighted.
Restore	Press to revert to data defined before the last change was made.
Report	Press to generate a report from the super table data.
OK	Press to save the super table for the project.
Cancel	Press to quit the routine.

Create Super Table

The Create Super Table option interfaces the definition of the HIP (horizontal intersection point) of each curve in the horizontal alignment with the selected design template(s), to create a Super Elevation Table for that curve.

The templates must be defined and allocated to the design using the Templates option in the Design menu BEFORE using the Create Super Table option. This is an essential step because the Create Super Table option reads the crossfall and width of the leg from the allocated template. There is no need to have vertical longitudinal design completed at this stage.

To create a super elevation table for the road, click **Design ▶ Create Super Table**. The *Create Super Table* table appears (Figure 18-68).

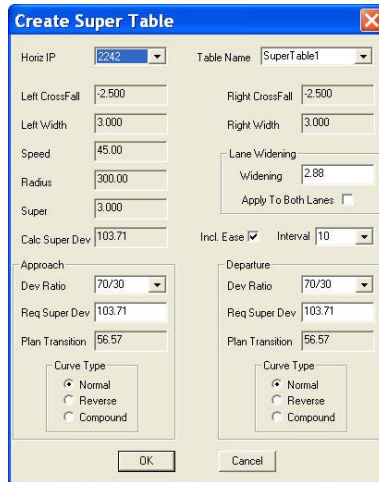


Figure 18-68. Create a Super Table

Table 18-17 describes the Create Super Table menu options.

Table 18-17. Create Super Table Menu Options

Field Type	Field Descriptions
Horiz IP	Select an HIP point number for the curves in the horizontal alignment from the drop-down box. NOTE: Each horizontal IP in the alignment is identified by the relevant point number in Survey View.
Table Name	Change the Default name by selecting an appropriate name, such as the name of the Curve; for example, use Curve1 for the first curve.
Left/Right Crossfall	The crossfall for the Left/Right Leg of the template.
Left/Right Width	The width for the Left/Right Leg of the template.
Speed	The Design Speed defined for the HIP
Radius	The Radius of the curve defined for the HIP
Super	The Superelevation defined for the HIP
Calc Super Dev (Superelevation Development Length)	The distance from the start of superelevation (SS) through to the maximum superelevation (E).

Table 18-17. Create Super Table Menu Options (Continued)

Field Type	Field Descriptions
Incl Ease/ / Interval	<ul style="list-style-type: none"> • Leave this box unchecked, unless required by local road design standards. • If box is check marked, select an Interval for this data.
Lane Widening	<ul style="list-style-type: none"> • Widening – A calculated value displays for suitable widening of the template leg through the curve. Edit as necessary or set to 0.000 for no widening. • Apply To Both Lanes – Check mark this box to widen both lanes through the curve. Leave unchecked to widen only the inside leg of the curve.
Approach	<p>This data defines the superelevation changes going into the curve. The fields depend on whether the curve is a Plain Curve or a Spiral Curve.</p> <ul style="list-style-type: none"> • Spiral Length – The spiral length applied to the curve design in Survey View. Only displays for Spiral Curves NOTE: For a Plain Curve, the Spiral Length field changes to Dev/Ratio. • Dev/Ratio – This ratio defines the percentage of the superelevation development length required prior to the tangent point approaching the curve and that required past the tangent point in the body of the curve. For closely spaced or common curves, all or most of the development will be within the curve at the departure of the first curve and the approach of the second curve. • Req Super Dev – Enter the required superelevation development length for the approach to the curve. This is usually computed in the <i>Calc Super Dev</i> field. • Plan Transition – Also known as the pavement rotation length.

Table 18-17. Create Super Table Menu Options (Continued)

Field Type	Field Descriptions
Curve Type	<ul style="list-style-type: none"> • Normal – if this box is check marked, the curve will be positioned in the alignment so that the approach to the curve starts from the standard crossfalls defined in the template, and rotates to the maximum superelevation. • Reverse – is this box is check marked, the curve will be positioned in the alignment so that the curve starts from the end of a previous curve and reverses direction. All the development is within the curve. The curve starts from the 0% tangent point position and builds up to maximum superelevation. NOTE: Select Reverse to set the Dev/Ratio field for a plain curve. • Compound – if this box is check marked, the approach to this curve follows a previous curve in the same direction, with insufficient room between curves to return to the standard crossfall of the template. NOTE: Usually not recommended by the design rules of Roading authorities, but may be necessary under some circumstances.
Departure	<p>Defines the superelevation changes leaving the curve. The fields will depend on whether the curve is a Plain Curve or a Spiral Curve. NOTE: For a Plain Curve the <i>Spiral Length</i> field is replaced with the <i>Dev/Ratio</i> field. For <i>Spiral Length</i>, <i>Dev/Ratio</i>, <i>Req Super Dev</i>, and <i>Plan Transition</i>, see the APPROACH field.</p>

Table 18-17. Create Super Table Menu Options (Continued)

Field Type	Field Descriptions
Curve Type	<ul style="list-style-type: none"> • Normal – if this box is check marked, the curve is positioned in the alignment so that the departure from the curve rotates from the maximum superelevation back to the standard crossfalls defined in the template. • Reverse – if this box is check marked, the curve is positioned in the alignment so that it joins to the next curve, which reverses direction. All the development is within the curve. The curve rotates from the maximum superelevation and ends at the 0% tangent point position. NOTE: Select Reverse to set the <i>Dev/Ratio</i> field for a Plain Curve. • Compound – if this box is check marked, departure from this curve allows insufficient room to return to the standard crossfall of the template, before the next curve, which turns in the same direction.
OK	Press to create the Super Table.
Cancel	Press to quit the routine.

Complete the definition of the superelevation for the curve, making sure the Table Name is chosen. The table may be edited as needed. The crossfall and widening changes are defined for the complete curve.



If the Super Table is correct, return to the Create Super Table option and recompute the Super Table.



Data is also written into the Super Table Selection dialog box to define the chainage at which the superelevation is applied.

Compound Side Slopes

The Compound Side Slopes option allows you to enter details of multiple and varying side slope legs to form compound side slopes. Each page or sheet of side slope information holds the detail for one leg of the side slope as it occurs in either a cut or a fill situation. Up to four different side slope solutions can be defined on each sheet, of which only one will be applied. The side slope chosen at each cross section depends on the depth to the natural surface compared to the calculated depth of the side slope. An unlimited number of batter legs can be defined on multiple sheets.



Data entered in the Compound Side Slope Definition menu is available to any road in the current project. Side slope sheets are applied to the design in the Compound Side Slope option.



Compound Side Slope Definitions that meet standard specifications may be entered in the similar option on the Library menu. A sheet held in the library may be copied **From** the library into a project, so that it may be modified to suit the project, or a sheet may be copied **To** the library for use in other projects. Library sheets may also be used directly in the road design from the list in the Compound Side Slope option.

To define compound side slopes, click **Design ▶ Compound Side Slopes Definition**. The *Compound Side Slopes* table appears (Figure 18-69 on page 18-79).

Figure 18-69. Compound Side Slopes

Table 18-18 describes the Compound Side Slope Definition table menu options.

Table 18-18. Compound Side Slope Definition Menu Options

Field Type	Field Definition
Compound Side Slope	You can change the \$\$DEFAULT name for the compound side slope displayed in the drop-down list. This data relates to one leg of the compound side slope. <ul style="list-style-type: none"> Click the New button to create a compound sideslope and allocate a name to this sheet or accept the default names, i.e., Sheet1, Sheet2, etc.
Cut/Fill	Each section has four rows for defining four varying side slopes, dependent on the depth to the natural surface compared to the calculated depth of the side slope.
Depth	Enter the depth below the natural surface (cut) or above the natural surface (fill) to which this side slope definition will apply.
Horizontal Distance	Enter the horizontal component of the side slope leg. This is usually positive, but could be negative if defining a bridge structure.
Vertical Distance	Enter the vertical component of the side slope leg. Enter a positive value, even in a fill situation. A negative value may be used to force a leg in the opposite direction to the norm. This field is not used if the 'Fixed Width' type of side slope is selected.
End Depth	Enter the required end depth. Only used with 'Normal' and "Continue' types of side slopes.

Table 18-18. Compound Side Slope Definition Menu Options (Continued)

Field Type	Field Definition
Normal	If selected, the side slope is stopped if it intersects the natural surface.
Intersect	If selected, the side slope continues at the specified grade until the natural surface is intersected.
Fixed Width	If selected, the side slope intersects the natural surface at the width specified.
Fixed Depth	If selected, the previous leg of the side slope will be forced to end a fixed depth from the natural surface. The depth from the surface is taken from the value specified in the Depth field. An intersecting final side slope leg is defined by the horizontal and vertical distance values entered.
Continue	If selected, the side slope continues for the horizontal distance specified. The side slope will continue through the natural surface, if necessary, to complete the leg.
New	Press to create a new sheet for the definition of one leg of a compound side slope. Default names will be Sheet1, Sheet2, and so on, but this name may be edited.
CopyFromLib	Press to display the list of compound side slopes definition sheets held in the library. Select a library sheet. It will be copied to the current sheet in the project, overwriting any existing data.
CopyToLib	Press to copy the current sheet to the Compound Side Slope library, using its defined name. If that name already exists in the Library, you are prompted to overwrite it.
OK	Press to save the sheets for the project.
Cancel	Press to quit the routine.

Each sheet is set up to represent one leg of the side slope, with options that will apply for different situations. Each row, marked 1-4, relates to a different side slope solution for this leg, dependent on the depth to the existing surface.

Symbol Table

The Symbol Table option defines symbols that may be attached to particular feature codes on a cross section plot. The symbol is selected from the list of symbols in the Symbol Library.

To define symbols for plotting, click **Design ▶ Symbol Table**. The *Symbol Table* appears (Figure 18-70).

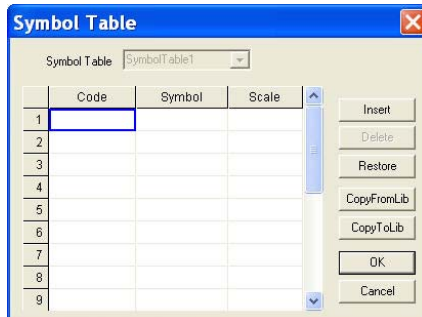


Figure 18-70. Define a Symbol Table

Table 18-19 describes the Symbol Table table menu options.

Table 18-19. Symbol Table Menu Options

Field Type	Field Descriptions
Symbol Table	The Default table name from the Library displays. NOTE: ONLY have table is defined in a project.
Code	Enter a point code used in the project; the symbol is attached to points with the specified code in the cross section plot.
Symbol	Select a symbol from the library symbol drop-down list.
Scale	Enter the scale required for the symbol on the plan. Most symbols are created at a millimeter size on a drawing. This option allows you to modify this size on the plot.
Insert	Press to create a new row from a highlighted row existing in the table.
Delete	Press to remove a highlighted row existing in the table. Row must be highlighted to activate the Delete button.
Restore	Press to revert to data defined before changes were made.
CopyFromLib	Press to display the symbol tables list in the project, adding data to any existing data.

Table 18-19. Symbol Table Menu Options (Continued)

Field Type	Field Descriptions
CopyToLib	Press to copy the current symbol table to the Symbol Table Library, using its defined name. If that name already exists in the Library, you are prompted to overwrite it.
OK	Press to save the symbol table for the project. It will be available for all roads within the project.
Cancel	Press to quit the routine.

Template Selection

The Template Selection option allows you to define a template to create the design surface of a road at specified chainage ranges. Templates defined in either the project or the Library can be selected



Template Selection does not define the subgrade surfaces, superelevation, side slopes or table drains; these are specified separately.

To specify where templates are applied, click **Design ▶ Template Selection**. The *Template Selection* dialog box appears (Figure 18-71).

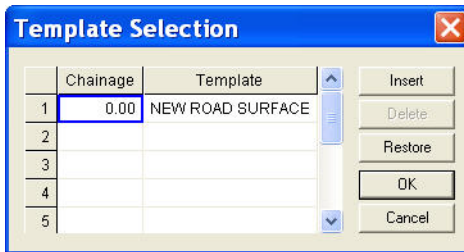


Figure 18-71. Defining Template Selections

Table 18-20 describes the Template Selection menu options.

Table 18-20. Template Selection Menu Option

Field Type	Field Descriptions
Chainage	Enter the chainage at which a template is applied. Each row defines the start of another template.

Table 18-20. Template Selection Menu Option (Continued)

Field Type	Field Descriptions
Template	Select the template from the drop-down entry box. Templates in the current project are listed in blue text and those in the Library are listed in black text.
Insert	Press to create a new row from a highlighted row existing in the table.
Delete	Press to remove a highlighted row existing in the table. Row must be highlighted to activate the Delete button.
Restore	Press to revert to data defined before changes were made.
OK	Press to save the data and exit.
Cancel	Press to quit the routine.

Super Table

The Super Table option allows you to create a super table that will be applied to modify a road template. These change are made to either widen a leg of the template or to modify the crossfall (grade) of a leg of the template through a curve. In many cases, both parameters will be changed. Super Tables defined in either the project or the Library can be selected.



The Super Table option only modifies the template legs defined in the Super table from the chainage where the super table is applied until the changes are completed.

To specify where super tables are applied, click **Design ▶ Super Table Selection**. The *Super Table Selection* table appears (Figure 18-72 on page 18-84).



Figure 18-72. Super Table

Table 18-21 describes the Super Table Selection table menu options

Table 18-21. Super Table Selection Menu Options

Field Type	Field Descriptions
Station (Start)	Enter the chainage at which a super table is applied. Each row define the start of another super table. The chainage entered relates to the running distance on the road where a superelevation change table begins. This is usually the point where the SS (0.00 distance) is defined in the super table.
Station (Reverse)	Enter the chainage where a super table is to be reversed. This is only used when the super table defines only the changes up to the maximum superelevation for a symmetrical curve. The chainage relates to the end of the maximum superelevation where the crossfalls reverse to their standard settings. NOTE: If no reversal is required, leave this field blank.
Super Table	Select the Super Table from the drop-down entry box. Super Tables in the current project are listed in blue text, while those in the Library are listed in black text.
Mirror	Check mark this field to mirror the changes defined in the Super Table. Do this to apply the change to a curve that turns in the opposite direction to that defined in the table. The left leg changes are applied to the right leg and vice-versa. NOTE: Leave this field unchecked if the template changes are applied to the curve exactly as defined in the Super Table.
Insert	Press to create a new row from a highlighted row existing in the table.
Delete	Press to remove a highlighted row existing in the table. Row must be highlighted to activate the Delete button.

Table 18-21. Super Table Selection Menu Options

Field Type	Field Descriptions
Restore	Press to revert to data defined before changes were made.
OK	Press to save the data and exit.
Cancel	Press to quit the routine.



The design surface is created using the changes to the template and display in the cross section pane when the design is computed.



- If Create Super Table is used to calculate the changes in a table, the start chainage will be automatically written into this selection table for the computer Super Table, so there is no need to enter further data in this table if Create Super Table is used for all the curves in the road design.
- If Super Tables are stored in the Library, it is usual to adopt a convention that defines the crossfall and width changes for a right-hand curve. When these are applied within a project, you can check mark the Mirror field to apply the Library Table to a left-hand curve.
- One or more lines in the table may be highlighted and then cut or copied and pasted to a new position. Right-click to display the context popup menu for these options.
- Data in the table may be saved to a text file or loaded from a text file. Click in the gray box at the top-left of the table and right-click with the mouse to use the option on the context popup menu.

Design Section Selection

The Design Section Selection option allows you to define various sections of an existing road surface, which are to be retained in the final design of road reconstruction work.

Each section is defined by a start and end chainage for its length and by a left and right string for its width. The strings must exist BEFORE this option can be used. Multiple sections of the existing road surface can be defined on subsequent rows of the dialog box.

To specify the sections of the natural surface included in the design surface, click **Design ▶ Design Section Selection**. The *Design Section Selection* table appears (Figure 18-73).

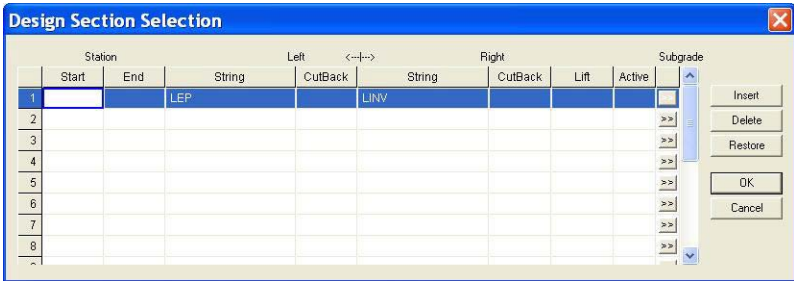



Figure 18-73. Design Section Selection

Table 18-22 describes the Design Section Selection table menu options.

Table 18-22. Design Section Selection Field Options

Field Type	Field Descriptions
Station (Start)	Enter the start chainage to define a chainage range within which the selected section will be applied as part of the design surface.
Station (End)	Enter the end chainage to complete a chainage range within which the selected section will be applied as part of the design surface.
Left (String)	Select a string from the drop-down entry box to define the left limit of the section. This string is usually a natural surface string and must already exist.
Left (CutBack)	Enter an option offset to be applied to the selected string so that the surface is cut back to a fixed distance inside the string.
Right (String)	Select a string from the pick list to define the right limit of the section.
Right (CutBack)	Enter an optional offset to be applied to the selected string so that the surface is cut back a fixed distance inside the string.
Lift	Enter a height difference in meters to be added to the selected section so that the final surface is raised by this amount to allow for an overlay.
Active	Check mark this field to activate the section defined in this row of data.

Table 18-22. Design Section Selection Field Options

Field Type	Field Descriptions
(Double Right Arrows) >>	<p>Press the double arrows button. The <i>Surface Record</i> dialog box displays (Figure 18-74).</p>  <p style="text-align: center;">Figure 18-74. Surface Record</p> <p>Use the following fields to define subgrade surfaces:</p> <ul style="list-style-type: none"> • Surface Name – elect a subgrade name from the drop-down entry box to define the depth of the sub-grade at the extents of the section defined in the main table. • Depth Left/Right – enter the depth to this surface from the left and right limits of the section defined in the main table.
Insert	Press to create a new row from a highlighted row existing in the table.
Delete	Press to remove a highlighted row existing in the table. Row must be highlighted to activate the Delete button.
Restore	Press to revert to data defined before changes were made.
OK	Press to save the data and exit.
Cancel	Press to quit the routine.

Using Design Section Selection This option could be used where a road is being reconstructed, with the addition of curb and channel to both sides of the existing bitumen seal. The design may require retention of the existing surface, extended to a new width where the curb and channel is built.

The left and right extents of the selected section are normally defined by the addition of strings to follow the existing edge of the seal, by clicking **Alignment ► Add String**. The string will usually be defined to follow the coded points along the existing seal edge (only if these

codes are in the natural surface cross section data), or from a string defined in the Survey View, or by setting an offset. It is common to cut back the existing seal edge by a small amount because the edges are broken or uneven. The *CutBack* field defines this option.

The design may require an overlay of the original bitumen seal to a depth of about 25mm and the *Lift* field should be used in this situation.

If subgrades are included in the design, then parameters for each subgrade surface must be set on the *Surface Record* dialog box (Figure 18-74 on page 18-87).



- Several sections of the existing road may be defined on different rows of the table.
 - This option should be used after completing the other components of the string line design.
 - One or more lines in the table may be highlighted and then cut or copied and pasted to a new position. Right-click to display the context popup menu for these options.
 - Data in the table may be saved to a text file or loaded from a text file. Click in the gray box at the top-left of the table and right-click with the mouse to use the option on the context popup menu.
-

Subgrade Template Selection

The Subgrade Template Selection option allows you to enter details of the templates applied to individual subgrade surfaces. These templates are often different for the design surface templates, especially if curbing is used.

To specify where templates are applied, and the depth required for each subgrade surface, click **Design ▶ Subgrade Template Selection**. The *Subgrade Template Selection* table appears (Figure 18-75 on page 18-89).

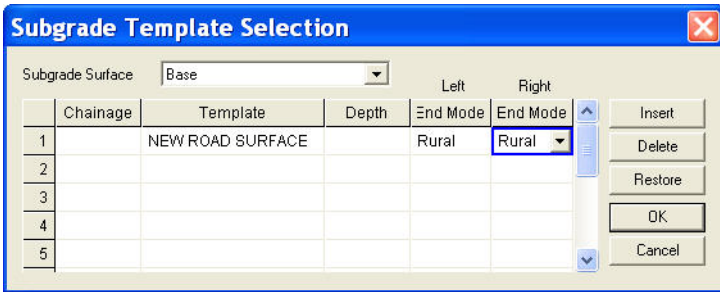


Figure 18-75. Subgrade Template Selection Table

Table 18-23 describes the *Subgrade Template Selection* dialog box menu options.

Table 18-23. Subgrade Template Selection Menu Options

Field Type	Field Descriptions
Subgrade Surface	Select a subgrade surface from the drop-down entry box to define their own template details. Each subgrade surface in the project will be listed in the drop-down menu.
Chainage	Enter the chainage at which a template is applied to the subgrade surface. Each row defines the start of another template.
Template	Select a template from the drop-down entry box. Templates in the current project are listed in blue text and those in the Library are listed in black text.
Depth	Enter the depth to the subgrade in meters; measured vertically below the design surface levels.

Table 18-23. Subgrade Template Selection Menu Options (Continued)

Field Type	Field Descriptions
Left (End Mode / End Mode)	<p>Controls the way the last leg of the subgrade intersects with either the design surface or the subgrade level on a string to the Left or right of the current string.</p> <p>Select either None, Rural, or Urban from the drop-down entry box.</p> <ul style="list-style-type: none"> • None – the end of the subgrade joins to the end of the same subgrade surface on the adjacent string. If no string is available, the subgrade joins vertically with the design surface. • Rural – the end of the subgrade continues at the crossfall of the last leg until it intersects with the design surface. NOTE: If this cannot be achieved in a cut situation, the leg continues out to the end of the design template, verge, or table drain and joins vertically with the design surface at this point. • Urban – the end of the subgrade turns vertically and intersects with the design surface at this point.
Insert	Press to create a new row from a highlighted row existing in the table.
Delete	Press to remove a highlighted row existing in the table. Row must be highlighted to activate the Delete button.
Restore	Press to revert to data defined before changes were made.
OK	Press to save the data and exit.
Cancel	Press to quit the routine.



- If the subgrade surfaces can be defined from a section of the design template copied down to the subgrade level, you should use the Pavement Parameters option in the Design menu, which removes the need to define templates and super tables for each subgrade surface.
- One or more lines in the table may be highlighted and then cut or copied and pasted to a new position. Right-click with the mouse to use the context popup menu options.
- Data in the table may be saved to a text file or loaded from a text file. Click in the gray box at the top-left of the table and right-click with the mouse to use the option on the context popup menu.

Subgrade Super Selection

The Subgrade Super Selection option allows you to apply superelevation table to modify the templates applied to the individual subgrade surfaces. In many cases, the data in the super table defined for the design surface may be duplicated for the subgrade surfaces, especially where only one leg of the template is involved.

To specify where super tables are applied to subgrade templates, click **Design ▶ Subgrade Super Selection**. The *Subgrade Super Selection* table appears (Figure 18-76 on page 18-92).



The settings for this dialog box are identical to those found in the Super Table Selection option (see Figure 18-72 on page 18-84), except that each subgrade surface is selected in turn, so that suitable entries may be made for it.

Subgrade Section Selection

The Subgrade Section Selection option allows you to define the section of an existing road surface that is retained as part of the subgrade surface or surfaces. This is usual when the road is being overlaid using the resheet options.

Each section is defined by a start and end chainage for its length and by a left and right string for its width.



The strings must exist BEFORE this option can be used.

To specify the sections of natural surface included in a subgrade surface, click **Design ▶ Subgrade Section Selection**. The *Subgrade Section Selection* table appears (Figure 18-76).



Figure 18-76. Subgrade Section Selection

Table 18-24 describes the Subgrade Section Selection table menu options.

Table 18-24. Subgrade Section Selection Menu Options

Field Type	Field Descriptions
Start Chainage	Enter the start chainage to define a chainage range within which the subgrades will use the existing surface.
End Chainage	Enter the end chainage to define a chainage range within which the subgrades will use the existing surface.
Left String	Select a string from the drop-down entry box to define the left limit of the section. This string is usually a natural surface string and must exist.

Table 18-24. Subgrade Section Selection Menu Options (Continued)

Field Type	Field Descriptions
Right String	Select a string from the drop-down entry box to define the right limit of the section.
Active	Check mark this field to activate the section defined in this row.
Insert	Press to create a new row from a highlighted row existing in the table.
Delete	Press to remove a highlighted row existing in the table. Row must be highlighted to activate the Delete button.
Restore	Press to revert to data defined before changes were made.
OK	Press to save the data and exit.
Cancel	Press to quit the routine.



- When Compute Design is run, the existing natural surface lying between the two string is used as part of each subgrade surface. Each subgrade surface will return to its specified depth at either side of this section.
- If a subgrade surface is above part of the selected section, it will be defined as a surface over the selected section until it intersects the natural surface. At this point, the natural surface will become part of the subgrade surface. This usually occurs only when considerable shape correction is being applied to the existing road.
- One or more lines in the table may be highlighted and then cut or copied and pasted to a new position. Right-click with the mouse to use the context popup menu.
- Data in the table may be saved to a text file or loaded from a text file. Click in the gray box at the top-left of the table and right-click with the mouse to use the options on the context popup menu.

Pavement Parameters

The Pavement Parameters option allows you to define a subgrade surface, based on a section of the template used for the surface above. Two labels on the template define the section. The subgrade surface also follows the higher surface through any superelevation, so you do

not need to set up templates or superelevation tables when this feature is used.

Pavement parameters can be set up for different design strings with templates attached to them, enabling multiple subgrade sections of a single cross section.

To define subgrade surfaces parallel to the surface above, click **Design ► Pavement Parameters**. The *Pavement Parameters* dialog box appears (Figure 18-77).

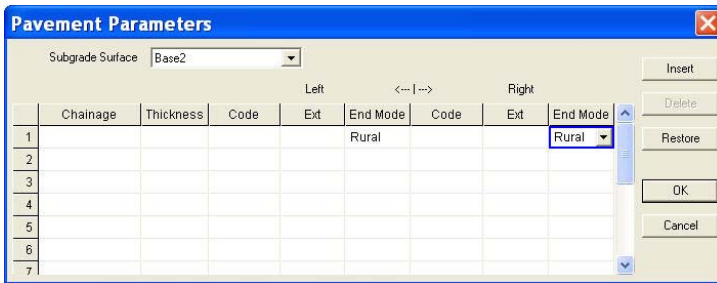


Figure 18-77. Pavement Parameters

Table 18-25 describes the Pavement Parameters table menu options.

Table 18-25. Pavement Parameters Menu Options

Field Type	Field Descriptions
Subgrade Surface	Select each surface in turn, commencing with the top subgrade in the sequence.
Chainage	Enter the chainage for the parameters on this row. If different parameters are required for a different segment of the design, enter data on a new row.
Thickness	Enter the thickness of this subgrade surface in meters. This is the thickness of this particular surface, unlike the settings for subgrade templates that are set at a depth from the design surface.
Left (Code)	Enter the code for the left extent of the template used for the subgrade. This will be a label used in the definition of one leg of the design surface template.

Table 18-25. Pavement Parameters Menu Options (Continued)

Field Type	Field Descriptions
Left (Ext)	If the pavement is to extend to the left of the template leg specified by the code, enter a distance in meters. Do not enter a negative value. NOTE: An extension might be needed to extend a leg underneath a curb with a vertical leg up to the design surface at the back of the curb, as shown in Figure 18-77 on page 18-94.
Left (End Mode)	Set the end mode to None, Rural, or Urban to define how the subgrade intersects the design surface at the left side of the design (see “Template Selection” on page 18-82 for further details).
Right (Code)	Enter the code for the right extent of the template used for the subgrade. This will be a label used in the definition of one leg of the design surface template.
Right (Ext)	If the pavement is to extend to the right of the template leg specified by the code, enter a distance in meters. Do not enter a negative value. NOTE: An extension might be needed to extend a leg underneath a curb with a vertical leg up to the design surface at the back of the curb, as shown in Figure 18-77 on page 18-94.
Right (End Mode)	Set the end mode to None, Rural, or Urban to define how the subgrade intersects the design surface at the right side of the design (see “Template Selection” on page 18-82 for further details)
Insert	Press to create a new row from a highlighted row existing in the table.
Delete	Press to remove a highlighted row existing in the table. Row must be highlighted to activate the Delete button.
Restore	Press to revert to data defined before changes were made.
OK	Press to save the data and exit.
Cancel	Press to quit the routine.

Enter the data for each subgrade surface in the order in which they are defined in the Subgrade Sequence on the Cross Section menu. The

subgrade surfaces are displayed in the cross section pane when Compute Design is run.



- One or more lines in the table may be highlighted and then cut or copied and pasted to a new position. Right-click with the mouse to use the context popup menu.
- Data in the table may be saved to a text file or loaded from a text file. Click in the gray box at the top-left of the table and right-click with the mouse to use the options on the context popup menu.

Side Slopes

The Side Slopes option allows you to specify side slopes. The ratios are defined as 1 for the vertical element and the value entered in the table as the horizontal element. A batter is a leg attached to the end of the last template leg (the hinge point) and pivots round this point to follow the slope specified at a particular chainage until the batter intersects with the natural surface.

This position will become the batter point and will be the extent of the design surface. These points are labeled LBAT and RBAT on the Design surface.

If you need the batter to finish at a particular offset from the centerline, the *Fixed Width Batters* option should be used instead.

To define sloping batters, click **Design ▶ Side Slopes**. The *Side Slopes* dialog box appears (Figure 18-78).

	Start Chainage	Left Cut	Left Fill	Right Cut	Right Fill
1	0.000	1.000	2.000	1.000	2.000
2	100.000	0.500	1.000	0.500	1.000
3	200.000	0.500	1.000	0.500	1.000
4	205.000	1.000	2.000	1.000	2.000
5					
6					
7					
8					
9					
10					

Figure 18-78. Side Slopes Table

Table 18-26 describes the Side Slopes table menu options

Table 18-26. Side Slope Menu Options

Field Type	Field Descriptions
Start Chainage	Enter the start chainage for the side slope settings defined on this row.
Left/Right Cut	Enter the horizontal component of the batter to be applied in most situations to the left and right ends of the template. Leave a field blank if no side slope is needed.
Left/Right Fill	Enter the horizontal component of the side slope to be applied in fill situations to the left and right ends of the template. Leave a field blank if no side slope is needed.
Insert	Press to create a new row from an existing highlighted row in the table.
Delete	Press to remove an existing highlighted row in the table. Row must be highlighted to activate the Delete button.
Restore	Press to revert to data defined before changes were made.
OK	Press to save the data and exit.
Cancel	Press to quit the routine.

Enter the data to define the side slopes used for this string. The side slopes are created and display in the cross section pane when Compute Design is run.

Using Side Slopes Side Slopes are interpolated between selected chainage ranges. In the above table, the batter slopes for any cross sections that lie between the 0.00m chainage and the 100.00m chainage will be clearly interpolated from the change in slopes between these two chainages (Figure 18-78 on page 18-96).

To hold the chainage at a constant slope, a row duplicating the settings at 100.00m chainage is entered at the 200.00m chainage. The side slopes will be the same for any cross sections that lie within this chainage range. Any cross section from 205.00m chainage to the end of the alignment uses the setting at 205.00m chainage.



- Side slopes are interpolated between selected chainage ranges.
- If you require batters to finish at a particular offset from the centerline, the Fixed Offset Slopes option should be used.
- If both Side Slopes and Offset Slopes are defined for the same chainage range, the Offset Slopes will be used in preference to the Side Slopes.
- One or more lines in the table may be highlighted and then cut or copied and pasted to a new position. Right-click with the mouse to use the context popup menu.
- Data in the table may be saved to a text file or loaded from a text file. Click in the gray box at the top-left of the table and right-click with the mouse to use the options on the context popup menu.

Fixed Offset Slopes

The Fixed Offset Slopes option allows you to specify offset slopes that intercept the natural surface at a given offset from the centerline. The offset slope intercept is entered in meters to the left or right of the centerline.

To define fixed offset slopes, click **Design ▶ Fixed Offset Slopes**. The *Fixed Offset Slopes* table appears (Figure 18-79).

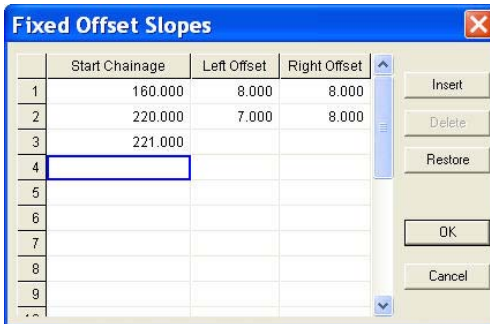


Figure 18-79. Fixed Offset Slopes

The table requires an offset to the left and to the right of the current string. This string may be the centerline of the road or a string defined for stringline design. In the latter case, you may only require a side slope at one side of the string, or no side slopes at all if there are strings to the left and right of it. The offsets are interpolated between the settings on one row and the next.

Enter the data to define the offset slopes and then click **OK** to save the data and exit. The offset slopes are created and display in the cross section pane when Compute Design is run.

Using Fixed Offset Slopes In Figure 18-79 above, the offset slope is created at the position 8m to the left and right of the string at chainage 160.00m and this offset is maintained at the right of the string for chainages 160.00m to 220.00m. Based on the way the dialog is filled out, the right offset will be 8' starting at station 160 and will continue to the end of the design. A fixed offset of 8' will be applied on the left beginning at 160, but will change to a 7' offset at 220 and will be maintained from that point up to the end of the design. A fixed offset for the start will also need to be defined. Typically you can define the fixed offset throughout the design, or you use the fixed slope option.



- Fixed Offset slopes are interpolated between selected chainage ranges.
 - If both Side Slopes and Fixed Width Slopes are defined for the same chainage range, the Fixed Width Slopes will be used in preference to the Side Slopes.
 - One or more lines in the table may be highlighted and then cut or copied and pasted to a new position. Right-click with the mouse to use the context popup menu.
 - Data in the table may be saved to a text file or loaded from a text file. Click in the gray box at the top-left of the table and right-click with the mouse to use the options on the context popup menu.
-

Compound Side Slopes

The Compound Side Slopes option allows you to define compound side slopes. You can build the side slope from the legs defined on individual sheets in the Compound Side Slope Definition option in the Design menu or the Compound Side Slopes in the Library menu.

A Compound Side Slope is a multi-leg side slope that can be used for such situations as benching in deep cut. This option is only used when side slope or fixed side slopes are not appropriate.

To define the components of a compound side slope, click **Design** ▶ **Compound Side Slopes**. The *Compound Side Slope Selection* table appears (Figure 18-80).

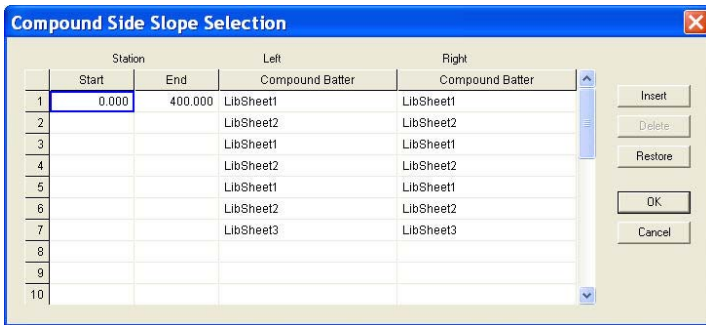


Figure 18-80. Compound Side Slopes

Table 18-27 describes the Compound Side Slopes table menu option.

Table 18-27. Compound Side Slopes Menu Options

Field Type	Field Descriptions
Chainage (Start)	Enter the start chainage of a chainage range for the compound side slope.
Chainage (End)	Enter the end chainage of a chainage range for the compound side slope.

Table 18-27. Compound Side Slopes Menu Options (Continued)

Field Type	Field Descriptions
Left (Compound Side Slopes)	Select the sheet or page to define the first leg of the compound side slope at the left side of the design. The second leg is set on the second row and subsequent legs on the third row onwards. The last leg would usually enable and intercept with the natural surface. NOTE: Leave the rows blank if no compound side slope is allocated at the left.
Right (Compound Side Slopes)	Select the sheet or page to define the first leg of the compound side slope at the right side of the design. The second leg is set on the second row and subsequent legs on the third row onwards. The last leg would usually enable an intercept with the natural surface. NOTE: Leave the rows blank if no compound side slope is allocated at the right.
Insert	Press to create a new row from a highlighted row existing in the table.
Delete	Press to remove a highlighted row existing in the table. Row must be highlighted to activate the Delete button.
Restore	Press to revert to data defined before changes were made.
OK	Press to save the data and exit.
Cancel	Press to quit the routine.

Define Compound Side Slopes, then press **OK** to save the data and exit the table. The side slopes are created and display in the cross section pane when Compute Design is run.

Strings are added to the list of strings for each batter leg (labeled LCB01, LCB02 and RCB01, RCB02, etc.)



- Check all cross sections to ensure that compound side slopes work correctly at each chainage.
- One or more lines in the table may be highlighted and then cut or copied and pasted to a new position. Right-click with the mouse to use the context popup menu.
- Data in the table may be saved to a text file or loaded from a text file. Click in the gray box at the top-left of the table and right-click with the mouse to use the options on the context popup menu.

Ditches

The Ditches option allows you to define water table drains. A drain is attached to the last leg of the road template. If a side slope is specified, the side slope is attached to the last leg of the ditch. The table drain can be defined with one leg, forming a V-shaped drain with the side slope leg, or with two legs, forming a U-shaped drain with the side slope leg.



Ditches are only applied to cross sections in a cut situation.

To define ditches, click **Design ▶ Ditches**. The *Ditches* table appears (Figure 18-81).

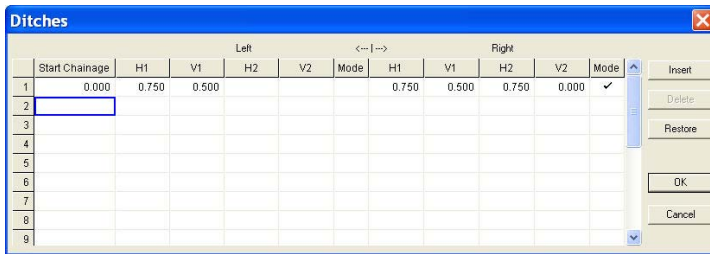


Figure 18-81. The Ditches Menu Option

Table 18-28 describes the Ditches table menu options.

Table 18-28. Ditches Field Descriptions

Field Type	Field Descriptions
Start Chainage	Enter the start chainage for the ditches defined on this row.
H1	Enter the horizontal component of the first leg of the drain in meters. Leave the field blank if no drain is required.
V1	Enter the vertical component of the first leg of the drain in meters. The end of this leg will be lower than, or equal to, the start of the leg. This is defined as a depth, so do not use a minus sign.
H2	Enter the horizontal component of the second leg of the drain in meters. Leave the field blank if no second leg is required.

Table 18-28. Ditches Field Descriptions

Field Type	Field Descriptions
V2	Enter the vertical component of the second leg of the drain in meters. The end of the leg will be higher than, or equal to, the start of the leg. Enter 0.00 for a leg that has no crossfall.
Mode	Used to define whether a drain should be created. Place check mark in this field to apply the ditch when the invert of the drain is below the natural surface level. Leave unchecked to apply the ditch when the end leg of the template is below the natural surface level.
Insert	Press to create a new row from a highlighted row existing in the table.
Delete	Press to remove a highlighted row existing in the table. Row must be highlighted to activate the Delete button.
Restore	Press to revert to data defined before changes were made.
OK	Press to save the data and exit.
Cancel	Press to quit the routine.

Ditches are created and display in the cross section pane when Compute Design is run.

Strings are added to the list of strings (labeled LT1, LT1, RT1, RT2) and the strings are shown in the alignment pane.



- Check all cross sections to ensure the ditches are correctly defined at each chainage.
- Ditches are interpolated between the settings on each row of the table.
- One or more lines in the table may be highlighted and then cut or copied and pasted to a new position. Right-click with the mouse to use the context popup menu.
- Data in the table may be saved to a text file or loaded from a text file. Click in the gray box at the top-left of the table and right-click with the mouse to use the options on the context popup menu.

Shoulder Extension

The Shoulder Extension option allows you to define shoulder extensions. A shoulder extension is an extension of the last leg of the template which continues at the existing crossfall for a specified width. They are only applied to cross sections in a fill situation – the end of the template is in fill.

To define shoulder extensions, click **Design ▶ Shoulder Extension**. The *Shoulder Extension* table appears (Figure 18-82).



Figure 18-82. Shoulder Extension

Table 18-29 describes the Shoulder Extension table menu options.

Table 18-29. Shoulder Extension Menu Options

Field Type	Field Descriptions
Start Chainage	Enter the start chainage for the additional shoulder extensions defined on this row.
Left Width	Enter the width of the shoulder extension. The last left-hand leg of the template is extended by this width. Leave the field blank if no shoulder extension is required.
Right Width	Enter the width of the shoulder extension. The last right-hand leg of the template is extended by this width. Leave the field blank if no shoulder extension is required.
Plot	Double-click with the mouse to check mark this field to plot a leader with levels for the shoulder extensions on the cross sections. Leave unchecked to plot the line work only.

Table 18-29. Shoulder Extension Menu Options

Field Type	Field Descriptions
Insert	Press to create a new row from a highlighted row existing in the table.
Delete	Press to remove a highlighted row existing in the table. Row must be highlighted to activate the Delete button.
Restore	Press to revert to data defined before changes were made.
OK	Press to save the data and exit.
Cancel	Press to quit the routine.

The shoulder extensions are created and display in the cross section pane when Compute Design is run.

Shoulder extension strings are added to the list of strings (labeled LV and RV). The strings display in the alignment pane.



- Check all cross sections to ensure the shoulder extensions are correctly defined at each chainage.
- Shoulder extensions are interpolated between the settings on each row of the table.
- One or more lines in the table may be highlighted and then cut or copied and pasted to a new position. Right-click with the mouse to use the context popup menu.
- Data in the table may be saved to a text file or loaded from a text file. Click in the gray box at the top-left of the table and right-click with the mouse to use the options on the context popup menu.


Compute Design

The Compute Design option allows you to compute and store the design surface of the road. It should be used after any of the design parameters previously discussed are altered to update the design surface in the alignment and cross section panes, and to save the design – making ready for the next stage.

To computer the design, either click **Design** ► **Compute Design** or press the **Compute Road Design** button  to the far right of the screen.

The design surface is computed to reflect the new design parameters. The results display in the three panes of the Roads View.

To automatically run Compute Design after any design changes have been made, do the following:

- Press the **Hot Button**  , located at the bottom of the main screen. The *Road Display Setting* dialog box displays (Figure 18-83).
- On the *CrossSection* tab, check mark the *Automatically Recompute Design* box.

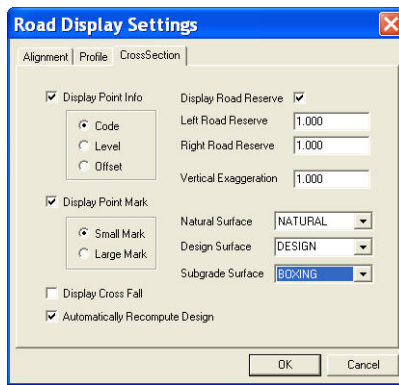


Figure 18-83. Automatically Recompute the Design for the Cross Section

Plotting Menu

This section describes the different Plotting menu commands located on the main toolbar (Figure 18-84).



Figure 18-84. Plotting Menu Options



To include volumes in a profile drawing, run the **Compute Volumes** command (**Cross Section ▶ Compute Volumes**) before plotting.

In This Section:

- “Profile Plot Settings”
- “Cross Section Plot Settings” on page 18-113
- “Plot Standard Profile” on page 18-120
- “Plot Offset Profile” on page 18-121
- “Plot CrossSection” on page 18-122

Profile Plot Settings

Use the parameters on the Profile Plot Settings tab to customize the parameters needed to create a cross section drawing by changing the parameters for cross section plot settings, surface plot settings, and cross section tables.

This option allows you to customize all of the parameters necessary to create a Profile drawing. There are six screens available, enabling you to specify parameters for General Plot Settings, Standard Profile Settings, Offset Profile Settings, Profile Table Settings, Cut/Fill Settings, and Volumes Settings.

Select the desired parameters on each screen that is relevant to your required drawing type (Offset or Standard Profile sections). Once

these are complete, execute one of the Plot commands in the menu to plot to the drawing editor. The Settings screens will also be available in the Drawing Editor.



To include volumes in a Profile Plot drawing, you will need to execute the Computer Volumes option **BEFORE** plotting.

To use the Profile Plot Settings command, do the following:

Click **Plotting** ► **Profile Plot Settings**. The *Profile Plot Settings* dialog box appears (Figure 18-85).



Figure 18-85. Profile Plot Settings

On the General Plot Settings tab, enter the following parameters:

1. Enter horizontal/vertical scale parameters for the profile, an X/Y coordinates for the start position, Datum parameters, and a title block drawing from the Library by clicking on the double arrows.
2. Check mark the Plot Grid box to enter grid data parameters and select the desired parameters for the plot.
3. When you have finished, click on another tab or click one of the following buttons:
 - **Load Default** – loads the previously saved defaults into input fields on all of the tabs in the project.
 - **Save Default** – saves the settings on all of the tabs as a default for future projects.
 - **OK** – saves your changes and closes the dialog box.

- **Cancel** – closes the dialog box without saving any changes.

On the **Standard Profile Settings** tab, enter the following parameters (Figure 18-86) to create and store standard parameters sets for plotting longitudinal sections:

1. Choose parameters for color, line type, thickness, and text style for either the Design Surface or the Natural Surface. Check mark the Active box for the surface to be included in the plot profile.
2. Check mark either the *Envelopes Max X-Fall* or *Min X-Fall* option and make the desired selections from the drop-down entry boxes.
3. Select a **Justification** table and a **Plot Sequence** from the drop-down entry boxes.

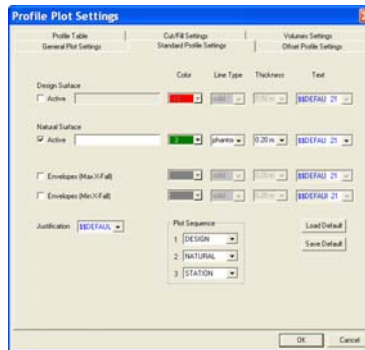


Figure 18-86. Plotting Menu – Standard Profile Settings

Fill out this screen if you want to plot a profile containing only the design and natural surface profiles of the current string.

4. When you have finished, click on another tab or click one of the following buttons:
 - **Load Default** – loads the previously saved defaults into input fields on all of the tabs in the project.
 - **Save Default** – saves the settings on all of the tabs as a default for future projects.
 - **OK** – saves your changes and closes the dialog box.
 - **Cancel** – closes the dialog box without saving any changes.



The Profile Table will be used for both Plot Standard Profile settings and Plot Offset Profile settings.

On the Offset Profile Settings tab, do the following to plot profiles of up to 8 different strings on the same figure (Figure 18-87):

1. Select as many offset profiles as desired by placing a check mark in the Active box and then selecting the string, surface, label, color type parameters.
2. Enter a value in the *Sep* column in order to space each of the profiles so that they do not cross each other and check mark Profile to plot the profile line and the levels on the long section. Uncheck this box to plot the levels on the long section but no profile line for the data defined on this row
3. Select a Title block file from the Library file by double-clicking the double arrows and then select Justification table from the drop-down list.

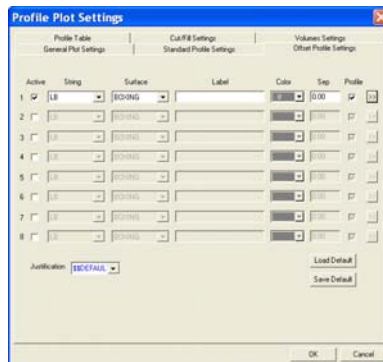


Figure 18-87. Plotting Menu – Offset Profile Settings

4. When you have finished, click on another tab or press one of the following buttons:
 - **Load Default** – loads the previously saved defaults into input fields on all of the tabs in the project.
 - **Save Default** – saves the settings on all of the tabs as a default for future projects.

- **OK** – saves your changes and closes the dialog box.
- **Cancel** – closes the dialog box without saving any changes.

On the Profile Table tab, do the following (Figure 18-88):

1. Enter parameters for **Horizontal lines**, **Grid/Leader Lines**, and **Line Spacing**.
2. Select a station format.
3. Select the text type and a label for the **Station**.
4. Select text style and **Justification** tables for **Stations**, **Datum**, and **IP Max/Min** fields. Enter a **Rounding** value for the **Datum**.
5. Select text style, justification tables, and line color for **VC Grade/Length** fields.
6. Select text style, line thickness, and line color for **Align** and **Super** fields.

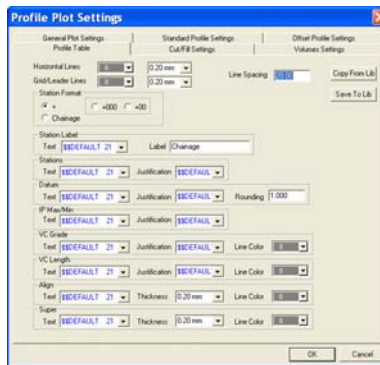


Figure 18-88. Plotting Menu – Profile Table Tab

7. Press **OK** to save or **Cancel** to quit and return to the main screen.

When you have finished, click on another tab or press one of the following buttons:

- **Copy From Lib** – selects previously designed tables from library for the profile plot setting.
- **Copy To Lib** – saves previously designed tables to the library for the profile plot setting.
- **OK** – saves your changes and closes the dialog box.

- **Cancel** – closes the dialog box without saving any changes.

On the Cut/Fill Settings tab, do the following to plot the level differences between nominated surfaces on the profile. Customize the screen to plot in two line format (Figure 18-89).

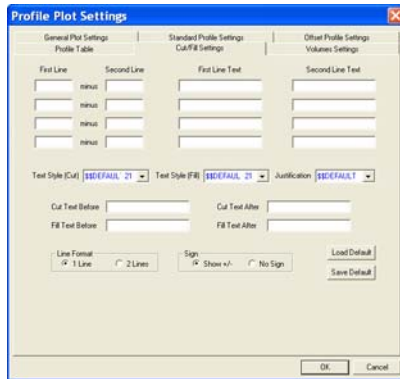


Figure 18-89. Plotting Menu – Cut/Fill Settings Tab



Cut/Fill settings can be specified on both Standard and Offset profiles.

When you have finished, click on another tab or click one of the following buttons:

- **Load Default** – loads the previously saved defaults into input fields on all of the tabs in the project.
- **Save Default** – saves the settings on all of the tabs as a default for future projects.
- **OK** – saves your changes and closes the dialog box.
- **Cancel** – closes the dialog box without saving any changes.

On the Volumes Settings, you can plot the previously computed volumes on the profile setting (Figure 18-90 on page 18-113).

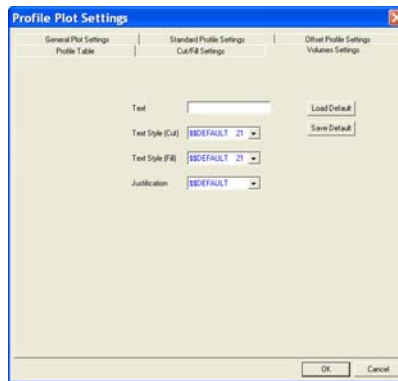


Figure 18-90. Plotting Menu – Volumes Settings Tab

When you have finished, click on another tab or click one of the following buttons:

- **Load Default** – loads the previously saved defaults into input fields on all of the tabs in the project.
- **Save Default** – saves the settings on all of the tabs as a default for future projects.
- **OK** – saves your changes and closes the dialog box.
- **Cancel** – closes the dialog box without saving any changes.

Cross Section Plot Settings

Use the parameters on the Cross Section Plot Settings tab to position the cross sections on a title block. Additional settings define the scale used, the extents and formats of the sections, and the required information included in each section.

To position the cross sections on a title block, do the following:

1. Click **Plotting** ► **Cross Section Plot Settings**.

The *Cross Section Plot Settings* dialog box appears (Figure 18-91 on page 18-114).

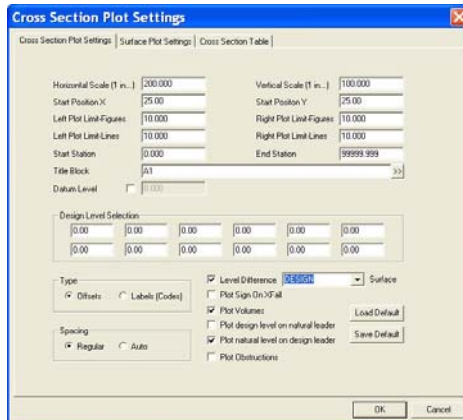


Figure 18-91. Cross Section Plot Settings – Cross Section Plot Settings Tab

- On the *Cross Section Plot Settings* tab, set the parameters as required in Table 18-30.

Table 18-30. Cross Section Plot Settings Field Descriptions

Field Name	Field Description
Horizontal Scale (1 in...)	Sets the horizontal scale for the cross sections.
Vertical Scale (1 in...)	Sets the vertical scale for the cross sections.
Start Position X	Sets the X coordinate of the starting position of the lower left corner of the first cross section on the page. This is measured in millimeters from the lower left corner of the title block.
Start Position Y	Sets the Y coordinate of the starting position of the lower left corner of the first cross section on the page. This is measured in millimeters from the lower left corner of the title block.
Left/Right Plot Limit Lines	Enter the offset limits for the plotting of offset and level figures. Enter an offset in meters to the left and right of the center line or string around which the sections are designed. Any data outside this limit is excluded from the cross sections.

Table 18-30. Cross Section Plot Settings Field Descriptions (Continued)

Field Name	Field Description
Left/Right Plot Limit Lines	Enter the offset limits for the plotting of line work. Enter an offset in meters to the left and right of the center line or string around which the sections are designed. Any data outside this limit is excluded from the cross sections.
Start Station	Enter the start chainage of the cross sections for this drawing. The default is 0.000 and can be used if the full length of the road will be plotted.
End Station	Enter the end chainage of the cross sections for this drawing. The default is 99999.999 and can be used if the full length of the road will be plotted.
Title Block	Select a suitable title block for the drawing from the library drop-down menu.
Datum Level	Enter a fixed datum level (in meters) in the adjacent field. The cross sections will be drawn relevant to this level on all of the sections, even when the grade of the road changes.
Design Level Selection	The format of the fields in this section depend on the option you choose in the Type field on this tab. They are only applied on rural format cross sections.
Type	<p>Offsets: The fields are set for the input of specified offsets. Enter a minus sign for offsets to the left of the center line. Levels will be interpolated from the design surface at the offsets and plotted on the cross sections at these positions for rural format cross sections.</p> <p>Labels (Codes): The fields are set for the input of labels or codes applied to the ends of particular legs of the cross section. Levels are interpolated from the design surface at these points and plotted on the rural format cross sections.</p>
Spacing	<p>Regular: Cross sections are positioned on the sheet by the row spacing set in the Cross Section Table, so this vertical spacing is applied from the datum line on the first section to the datum line of the next section.</p> <p>Auto: Cross sections are positioned on the sheet by the Auto row spacing set in the Cross Section Table, so this vertical spacing is applied from the highest point on the first section to the datum line of the next section.</p>

Table 18-30. Cross Section Plot Settings Field Descriptions (Continued)

Field Name	Field Description
Level Difference	Plots the height difference between the natural design surfaces at each offset on the full format cross sections. These are plotted in an extra row above the surface levels. Where the design surface is cut below the natural surface, the level difference is negative. Check mark this box, then select a surface type from the drop-down list.
Plot Sign on XFall	Plots the minus sign on the crossfall if the crossfall is activated on the Surface Plot Settings tab.
Plot Volumes	Plots the volume between this cross section and the previous one. The volumes must have been computed using the Compute Volumes on the Cross Section menu.
Plot Design Level on Natural Leader	Plots the design surface levels on the leader lines for the natural surface when a full format cross section is plotted. A design level is interpolated and plotted wherever there is a leader line relating to the natural surface. Clear this check box to plot the design levels only where the leader lines relate to the design surface.
Plot Design Level on Design Leader	Plots the natural surface levels on the leader lines for the design surface when a full format cross section is plotted. A natural surface level is interpolated and plotted wherever there is a leader line relating to the design surface. Clear this check box to plot the natural surface levels only where the leader lines relate to the natural surface.
Plot Obstruction	Plots any obstructions.
Load Default	Loads the previously saved defaults into input fields on all of the tabs in the project.
Save Default	Saves the settings on all of the tabs as a default for future projects.
OK	Saves your changes and closes the dialog box.
Cancel	Closes the dialog box without saving any changes.

Surface Plot Settings Use the *Surface Plot Settings* dialog box to define the plot parameters for the various surfaces included in the drawing. Select colors, text styles, line types, and number justification for any required surface using on this tab. Additional options allow you to label cross sections with cross falls and plot symbols on each section.

To define the plot parameters for the various surfaces included in the drawing, do the following:

1. Click **Plotting** ► **Cross Section Plot Settings**.

The *Surface Plot Settings* dialog box appears (Figure 18-92).

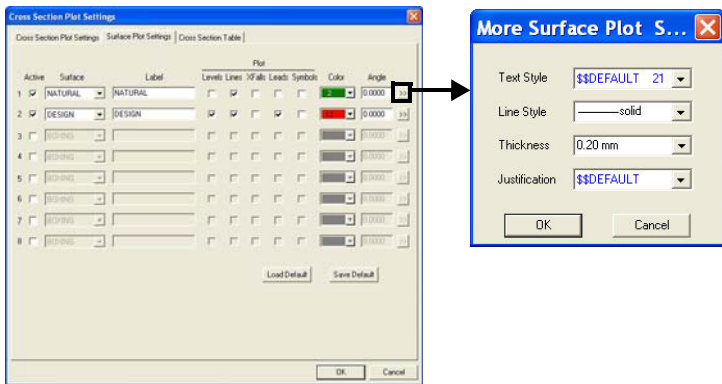


Figure 18-92. Cross Section Plot Settings – Surface Plot Settings Tab

2. Click on the *Surface Plot Settings* tab to set the parameters as required in Table 18-31 below.

Table 18-31. Surface Plot Settings Field Descriptions

Field Name	Field Description
Active	Check mark this box to include the corresponding surface in the cross section plot.
Surface	Select the Surface from the drop-down list to be included in the plot.
Label	Enter the surface name used to label rows of levels on the full format cross sections.

Table 18-31. Surface Plot Settings Field Descriptions (Continued)

Field Name	Field Description
Plot	<ul style="list-style-type: none"> • Levels: Check mark this box to include levels for the surface. Subgrades usually have this check box cleared. A row of levels is included in the full format cross sections. • Lines: Check mark this box to display the surface profile. The center line level is displayed for the surface for rural format cross sections. • XFalls: Check mark this box to display cross falls for the surface. • Leads: Check mark this box to display leader lines at leveled offsets. • Symbols: Check mark this box to display symbols.
Colour	Select a color of the profile line work from the drop-down entry box.
Angle	Enter a text angel for levels on an URBAN cross section.
>>	Press to display the <i>More Surface Plot Specifications</i> dialog box (see Figure 18-92 on page 18-117). Specify text tables, line styles, thickness, and justification for text.
Load Default	Loads the previously saved defaults into input fields on all of the tabs in the project.
Save Default	Saves the settings on all of the tabs as a default for future projects.
OK	Saves your changes and closes the dialog box.
Cancel	Closes the dialog box without saving any changes.

Cross Section Table Use the *Cross Section Table* tab to define the plot parameters for the cross section layout, including the text styles used. Use the Cross Section Table to define the appearance of the cross section format and details of each cross section, leaders, row and column spacing, horizontal line spacings, labels, text types, and colors.

This tab displays the default settings loaded from previously saved defaults. If the default settings are not suitable for this plot, you can copy a previously designed alternative layout from the library. To do

this, press the **Copy From Lib** button and select one of the previously defined tables.



A default template can be set up and saved to the Library. Note that Justification (Precision) and Text Styles can be edited and checked with the Settings option under the Survey menu.

To define the plot parameters for the cross section layout, do the following:

1. Click **Plotting ► Cross Section Plot Settings**.
On the *Cross Section Plot Settings* dialog box, click on the *Cross Section Table* tab (Figure 18-93).

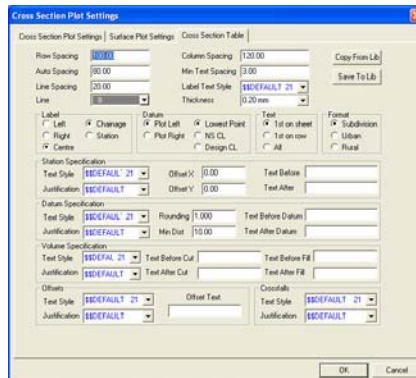


Figure 18-93. Cross Section Plot Settings – Cross Section Table Tab

2. Click on the *Surface Plot Settings* tab to set the parameters as required. For field descriptions, see the “Cross Section Field Descriptions” on page 18-26 command in the **Library** menu.
3. When you have finished, click on another tab or click one of the following buttons:
 - **Load Default** – loads the previously saved defaults into input fields on all of the tabs in the project.
 - **Save Default** – saves the settings on all of the tabs as a default for future projects.
 - **OK** – saves your changes and closes the dialog box.
 - **Cancel** – closes the dialog box without saving any changes.

When the parameters have been appropriately defined, a cross section drawing may be created in the Drawing View using the **Plotting ▶ Plot Cross Section** command.

Plot Standard Profile

Use the Plot Standard Profile command to plot a standard profile as a drawing in the Drawing View. A standard profile contains profiles of the natural surface and the design surface for the current string in the Cross Section Editor.

To use this command, click **Plotting ▶ Plot Standard Profile**. The Drawing View opens as the current view and a linked drawing is created (Figure 18-94), containing the sheet(s) that comprise the profile drawing.



The parameters defining the linked drawing are carried over into the Drawing View. You can modify these parameters by using **Settings ▶ Standard Profile Settings** in the Drawing View to fine-tune the drawing layout and apply modified parameters to suit the particular profile plot.

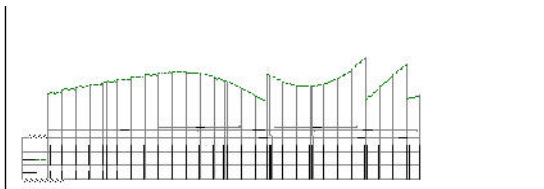




Figure 18-94. Plot a Standard Profile

If the design in the Cross Section Editor is modified, the linked drawing may be updated in the Drawing View using the **Regen**  icon. There is no need to run the **Plot Standard Profile** command again from the Cross Section Editor as this creates another drawing, losing any modifications already made in the Drawing View.

The title block(s) of a linked drawing may be edited by clearing (Unchecking) the **Locked** check box in the **Road Display Settings**

(Hot Button ) dialog box. A linked drawing may be saved as an unlinked drawing for any minor editing before plotting the result.


Plot Offset Profile


Use the Plot Offset Profiles command to plot an offset profile as a drawing in the Drawing View. An offset profile contains up to eight profiles along various strings and surfaces in the Cross Section Editor. The tabs in the *Profile Plot Settings* dialog box used to define the drawing are carried forward into the Drawing View.

To use the Plot Offset Profile command, click **Plotting ▶ Plot Offset Profiles**. The Drawing View opens as the current view and a linked drawing is created, containing the sheet(s) that comprise the offset profile drawing.



The parameters defining the linked drawing are carried over into the Drawing View. You can modify these parameters by using **Settings ▶ Offset Profile Settings** in the Drawing View to fine-tune the drawing layout and apply modified parameters to suit the particular profile plot.

If the design in the Cross Section Editor is modified, the linked drawing may be updated in the Drawing View using the **Regen**  icon. There is no need to run the **Plot Offset Profile** command again from the Cross Section Editor as this creates another drawing, losing any modifications already made in the Drawing View.

The title block(s) of a linked drawing may be edited by clearing the **Locked** check box in the **Road Display Settings** (Hot Button ) dialog box. A linked drawing may be saved as an unlinked drawing for any minor editing before plotting the result.

Plot CrossSection

Use the Plot CrossSection command to plot sheets of a cross section as a drawing in the Drawing View. The drawing is formatted depending on the settings made in the **Cross Section Plot Settings** command.

To use the Plot CrossSection command, click **Plotting ▶ Plot Cross Section**. The Drawing View opens as the current view and a linked drawing is created, containing the sheet(s) that comprise the cross section drawing (Figure 18-95).



The parameters defining the linked drawing are carried over into the Drawing View. You can modify these parameters by using **Settings ▶ Cross Section Settings** in the Drawing View to fine-tune the drawing layout and apply modified parameters to suit the particular profile plot.

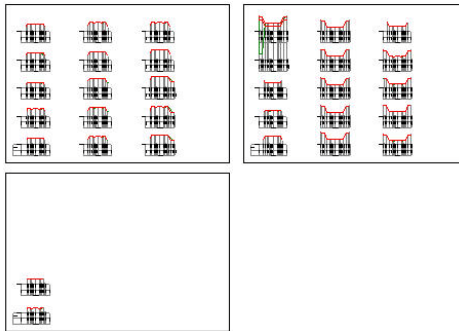




Figure 18-95. Plotting the Cross Section

If the design in the Cross Section Editor is modified, the linked drawing may be updated in the Drawing View using the **Regen**  icon. There is no need to run the **Plot Cross Section** command again from the Cross Section Editor as this creates another drawing, losing any modifications already made in the Drawing View.

Cross sections are usually plotted on multiple sheets created within the one drawing.

The title block(s) of a linked drawing may be edited by clearing (unchecking) the **Locked** check box in the **Display Settings** (Hot Button ) dialog box. A linked drawing may be saved as an unlinked drawing for any minor editing before plotting the result.

Library Menu

This section describes the different Library menu commands in Roads View (Figure 18-96).

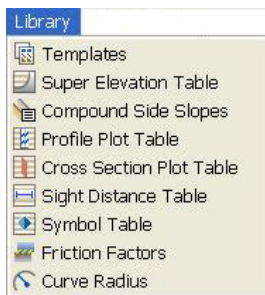


Figure 18-96. Roads View – Library Menu Option



The Templates, Super Elevation Table, Symbol Table, and Compound Slopes menu options are discussed under the Roads View: Design Menu chapter.

In This Section:

- “Profile Plot Table” on page 18-124
- “Cross Section Plot Table” on page 18-127
- “Sight Distance Table” on page 18-131
- “Friction Factors” on page 18-133
- “Curve Radius” on page 18-134

Profile Plot Table

Use the Profile Plot Table command to define the appearance of the profile layout in regard to grid line, leaders, horizontal line spacings, labels, text, types, and colors. Several tables may be defined in the library, so you can have different profile “parameterized” layouts for different types of projects.

When the *Profile Table* tab on the *Plot Settings* dialog box is opened, it displays the default settings loaded from previously saved defaults. You can copy a previously designed profile table form the library by clicking **CopyFrom Lib** on the *Profile Table* tab and then selecting one of the available tables.

To use the Profile Table command, do the following:

1. Click **Library ▶ Profile Table**.

The library file defined in the **Program Setup** of the Survey View appears with the first table displayed (Figure 18-97).

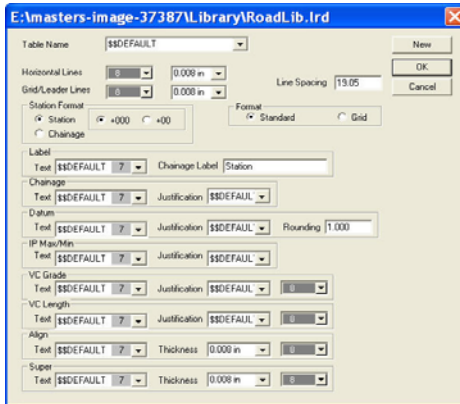


Figure 18-97. Library Menu – Profile Table

2. Set the parameters as required in Table 18-32.
3. When you have finished, click **OK** to save your changes and close the dialog box.

Table 18-32. Profile Table Field Descriptions

Field Name	Field Description
Table Name	This is the name of the profile table that displays.

Table 18-32. Profile Table Field Descriptions (Continued)

Field Name	Field Description
Horizontal Lines	Select the color and thickness required for the horizontal lines in the profile layout.
Grid/Leader Lines	Select the color and thickness required for the vertical leader lines and grid lines if used in the profile layout.
Line Spacing	Enter the spacing in millimeters between the horizontal lines in the profile layout.
Station/Chainage Format	Select either a station or a chainage format.
Format	Select wither a standard format (most common) or a grid format.
Label	<ul style="list-style-type: none"> • Text: Select the required text style for these labels from the drop-down entry box <p>Labels are used at the left end of the chainage and levels section of the profile layout to name the individual rows of data, such as chainage, natural surface, and alignment details.</p> <ul style="list-style-type: none"> • Chainage Label: Enter a name for the Chainage row. Either Chainage, Distance, or some other label.
Chainage	<p>These settings relate to the numerical chainage values plotted on the chainage row.</p> <ul style="list-style-type: none"> • Text: Select the required text style for the chainages or distances from the drop-down entry box. • Justification: Select the required Number Justification Table to define the format and precision of the numerical values from the drop-down entry box.
Datum	<p>These settings relate to the Datum value plotted on the left side of the datum line.</p> <ul style="list-style-type: none"> • Text: Select the required text style for the datum value from the drop-down entry box. • Justification: Select the required Number Justification Table to define the format and precision of the numerical values from the drop-down entry box. • Rounding: Enter a rounding increment for the datum value. This is usually set to 1.000 to round the datum value to a whole number from the drop-down entry box.

Table 18-32. Profile Table Field Descriptions (Continued)

Field Name	Field Description
IP Max/Min	<p>These settings relate to the maximum and minimum IP values plotted at the high and low points of the design profile line.</p> <ul style="list-style-type: none"> • Text: Select the required text style for the IP Max/Min value. from the drop-down entry box. • Justification: Select the required Number Justification Table to define the format and precision of the numerical values from the drop-down entry box.
VC Grade	<p>These settings relate to the vertical curve (VC) grade plotted below the design profile line .</p> <ul style="list-style-type: none"> • Text: Select the required text style for the VC Grade values from the drop-down entry box. • Justification: Select the required Number Justification Table to define the format and precision of the numerical values from the drop-down entry box. • Line Color: Select the color for the horizontal line plotted to define the extent of each grade from the drop-down entry box.
VC Length	<p>These settings relate to the vertical curve (VC) length plotted below the design profile line.</p> <ul style="list-style-type: none"> • Text: Select the required text style for the VC Length values from the drop-down entry box. • Justification: Select the required Number Justification Table to define the format and precision of the numerical values from the drop-down entry box. • Line Color: Select the color for the horizontal line plotted to define the length of each vertical curve from the drop-down entry box.
Align	<p>These settings relate to the horizontal alignment details plotted in the profile layout.</p> <ul style="list-style-type: none"> • Text: Select the required text style for the alignment details. • Thickness: Select a thickness for the line work used. • Line Color: Select the color for the line work plotted to define the alignment details.

Table 18-32. Profile Table Field Descriptions (Continued)

Field Name	Field Description
Super	<p>These settings relate to the Super Elevation Details plotted in the profile layout.</p> <ul style="list-style-type: none"> • Text: Select the required text style for the Super Elevation Details from the drop-down entry box. • Thickness: Select a thickness for the line work used from the drop-down entry box. • Line Color: Select the color for the line work plotted to define the Super Elevation Details from the drop-down entry box.

You can edit the current table for future profile plots or create a new table.

To create a new table, do the following:

1. Click **New**, and enter a name for the new table.
2. Input the required parameters, and then click **OK** to save the table.

Cross Section Plot Table

The Cross Section Plot Table command allows you to define standard parameters for plotting sheets of cross sections. The parameter stored in the Cross Section Plot Table are selected to produce a suitable format for the Cross Section drawings.

The Cross Section Table is used to define the appearance of the Cross Sections with respect to the preferred layout, horizontal lines, leaders, horizontal line spacings, labels, text types, and colors. Several tables can be defined in the Library so that different cross section parametrized layouts for different types of projects are available.

To create cross section plot layouts, click **Library ▶ Cross Section Plot Table**. The *roads library cross section plot table* appears (Figure 18-98 on page 18-128).

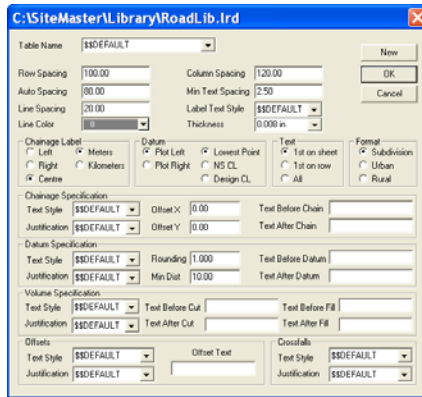


Figure 18-98. Roads View: Cross Section Plot Table

Table 18-33 describes the Cross Section Plot Table field options.

Table 18-33. Cross Section Plot Field Options

Field	Description
Table Name	The Default name of the table displays. Press New to create a new table.
Row Spacing	Enter the spacing (in millimeters) for the Regular spacing mode set in the Cross Section Plot Settings screen. This is the distance between the datum line of one cross section and the datum line of the next cross section in the row above.
Column Spacing	Enter the spacing (in millimeters) between adjacent columns of cross sections on the drawing.
Auto Spacing	Enter the spacing (in millimeters) for the auto spacing mode set in the Cross Section Plot Settings screen. This is the distance between the highest point on one cross section and the datum line of the next cross section in the row above.
Min Text Spacing	Enter the minimum spacing between the leader lines (in millimeters) This setting controls the spacing required for the text between the leader lines.
Line Spacing	Enter the spacing (in millimeters) between the horizontal lines in the full/urban format cross section layout. This is the spacing for the offset and level details.

Table 18-33. Cross Section Plot Field Options (Continued)

Field	Description
Label Text Style	Select the font style desired for the labels that name the lines of offsets and levels.
Line Color	Select a color for the horizontal lines used.
Thickness	Select a line thickness for the horizontal lines used.
Chainage Label	<p>Numerical chainage values plotted to label each cross section:</p> <ul style="list-style-type: none"> • Select either Left, Right, or Center to position the Chainage label of each cross section. • Select either Meters or Kilometers for the unit of measurement for the Chainage label.
Datum	<p>Select the following parameters for the datum label plotted beside the datum line:</p> <ul style="list-style-type: none"> • Select either Plot Left or Plot Right for each cross section. • Select one of the following to position the datum line relative to cross section data. <ol style="list-style-type: none"> 1. Lowest Point: the datum distance is at the lowest point of the cross section. 2. NS CL: the datum distances is at the centerline point on the design surface of the cross section. 3. Design CL: the datum distance is at the centerline point on the design surface of the cross section <p>NOTE: The final result depends of this setting depends on the settings in the Datum Specification fields.</p>
Text	<p>Select parameters for the labeling of the rows of numerical data on the full format / urban cross sections.</p> <ul style="list-style-type: none"> • 1st on sheet: the surface labels and offset label are only on the first cross section at the bottom-left corner of each sheet. • 1st on row: The surface labels and offset label are only on the first cross section at the left hand end of each row of sections. • All: The surface labels and offset label are only on the first cross section at the left hand end of each row of sections.

Table 18-33. Cross Section Plot Field Options (Continued)

Field	Description
Format	<p>Set the radio button to define the format of the cross sections:</p> <ul style="list-style-type: none"> • Subdivision: offsets and levels are plotted within a framework of horizontal and vertical lines for each cross section. This is the Full format cross section. • Urban: cross sections are plotted in the same format as above except that the offsets are not plotted. • Rural: levels are only shown at the centerline point and there is no framework of lines, just a horizontal datum line and a vertical line at the centerline.
Chainage Specification	<p>Enter the following parameters for the chainage labels:</p> <ul style="list-style-type: none"> • Text Style: select a text style • Justification: select a number justification style. • Offset X/Y: enter a positive or negative value (in millimeters) to move the chainage labels to a modified position. The offsets will move the label relative to the setting made for the chainage label (Left, Right, or Center). • Text Before/After Chain: enter any required text before and after the chainage label.
Datum Specification	<p>Enter the following parameters for the datum labels:</p> <ul style="list-style-type: none"> • Text Style: select the desired text style. • Justification: select the desired number justification style. • Rounding: enter a rounding increment for the datum value, usually set to 1.000 to round the datum value to a whole number. • Min Dist: enter a value (in millimeters) to set the minimum distance between the datum line and the position set for this distance in the Datum option above. • Text Before/After Datum: enter any required text before and after the datum label.

Table 18-33. Cross Section Plot Field Options (Continued)

Field	Description
Volume Specification	Enter the following parameters for the volumes text, if plotted, on each cross section: <ul style="list-style-type: none"> • Text Style: select the desired text style. • Justification: select the desired number justification style. • Text Before/After Cut: enter any text required for cut volumes. • Text Before/After Fill: enter any text required for fill
Offsets	Enter the following parameters for the Offsets text, if plotted, on each cross section: <ul style="list-style-type: none"> • Text Style: select the desired text style for the offsets label. • Justification: select the desired number justification style for the offsets label.
Offset Text	Enter the label for the Offsets row on the full format cross section.
Crossfalls	Enter the following parameters for the crossfalls text, if plotted, on each cross section: <ul style="list-style-type: none"> • Text: select the desired text style for the crossfalls text. • Justification: select the desired number justification style for the crossfalls text.

Sight Distance Table

The Sight Distance Table command is used when editing a Vertical IP to create a vertical curve that is based on sight distance criteria from the applicable standards, such as Austroads or the RTA “Road Design Guide”. The user can set up multiple tables in the library based on local standards.

To set parameters in the Sight Distance Table, click **Library ▶ Sight Distance Table**. The *roads library sight distance table* appears (Figure 18-99 on page 18-132).

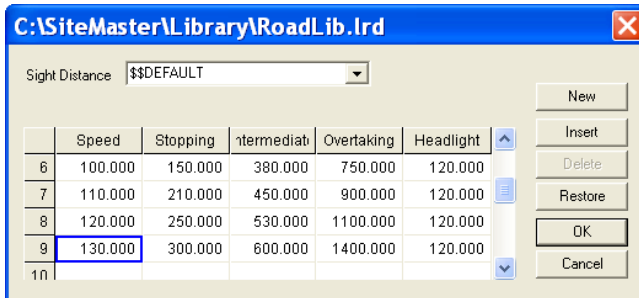


Figure 18-99. Roads View – Sight Distance Table

Table 18-34 describes the Sight Distance Table field options.

Table 18-34. Sight Distance Table Field Options

Field	Description
Sight Distance	The Default name of the Sight Distance table currently on display.
Speed	Enter the Design Speed for the parameters on this row of data.
Stopping	Enter the visibility required (in meters) to see an object in the lane of travel in order to stop before striking it.
Intermediate	Enter a value (in meters) that enables a driver to travel a road in comfort with reasonably safe overtaking opportunities. NOTE: This is used if the Overtaking sight distance provision is prohibited by cost.
Overtaking	Enter a value (in meters) that provides sufficient visibility to detect oncoming vehicles in time to allow safe overtaking of a vehicle moving below the speed limit.
Headlight	Enter a value (in meters) that enables a driver to spot a stopped vehicle. This is usually only used at low speeds.
New	Press to create a new table; enter a name for the new sight distance table, or accept the default name.
Insert	Press to insert a new row in the sight distance table.
Delete	Press to delete a highlighted row in the sight distance table.
Restore	Press to restore a deleted row in the sight distance table.
OK	Press to save the table parameters.
Cancel	Press to quit the routine.

Friction Factors

The Friction Factors table is used when editing a Horizontal IP, to define the maximum design value of side friction for the type of pavement surface, at specific design speeds.

The table will be set up for applicable standards, such as Austroads or the RTA “Road Design Guide”. You can set up multiple tables in the library, based on local standards.

To set Friction Factors parameters, click **Library ▶ Friction Factors**. The *roads library friction factors* table appears (Figure 18-100).

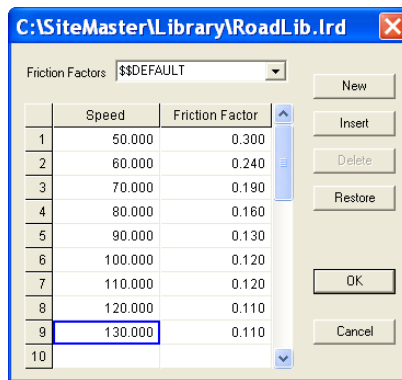


Figure 18-100. Roads View – Friction Factors Table

Table 18-35 describes the Friction Factors field options.

Table 18-35. Friction Factors Field Options

Field	Description
Friction Factors	The default name of the Friction Factors table currently on display.
Speed	Enter the Design Speed.
Friction Factor	Enter the friction factor for the relevant design speed.
New	Press to create a new table; enter a name for the new Friction Factors table.
Insert	Press to insert a new row in the friction factors table.
Delete	Press to delete a highlighted row in the friction factors table.
Restore	Press to restore a deleted row in the friction factors table.

Table 18-35. Friction Factors Field Options (Continued)

Field	Description
OK	Press to save the table parameters.
Cancel	Press to quit the routine.

The Friction Factors table is used when editing a horizontal intersection point (IP) of an alignment in Road View. The required friction factor is used for the selected design speed when computing an appropriate curve radius superelevation.

Curve Radius

The Curve Radius command is used when editing a Horizontal IP, to define the minimum curve radius at specified design speeds.

The table will be set up for the applicable standards, such as Austroads or the RTA “Road Design Guide”. You can set up multiple tables in the library, based on local standards.

Variables are defined and applied for a specified Design Speed when computing the minimum radius for the curve.

To define the curve radius, click **Library ▶ Curve Radius**. The *roads library curve radius* table appears (Figure 18-101).

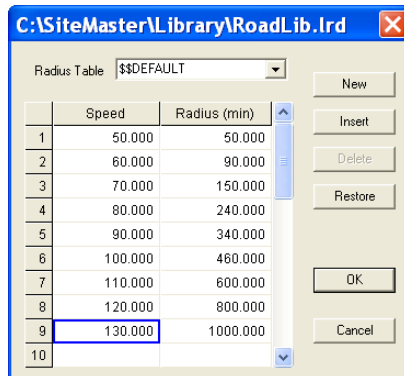


Figure 18-101. Roads View – Curve Radius

Table 18-36 describes the Curve Radius field options.

Table 18-36. Curve Radius Field Options

Field	Description
Radius Table	The default name of the Curve Radius table currently on display.
Speed	Enter the Design Speed.
Radius (min)	Enter the minimum radius (in meters) for the design of the curve at the selected speed.
New	Press to create a new table; enter a name for the new curve radius table.
Insert	Press to insert a new row in the curve radius table.
Delete	Press to delete a highlighted row in the curve radius table.
Restore	Press to restore a deleted row in the curve radius table.
OK	Press to save the table parameters.
Cancel	Press to quit the routine.

Window Menu

The Window Menu in the Cross Section Editor is similar to the Window Menu in the Survey View. For information about this menu, see “New Window” on page 17-2.

Help Menu

The Help Menu in the Cross Section Editor is similar to the Help Menu in the Survey View. For information about this menu, see “Help Menu” on page 17-4.



Topcon Positioning Systems, Inc.

7400 National Drive, Livermore, CA 94550
800-443-4567 www.topconpositioning.com



ISO 9001:2000
FM 68448

MAGNET Office Reference Manual

P/N: 1001299-01 Rev A 07/12

©2012 Topcon Corporation All rights reserved. No unauthorized duplication.